CONTENTS

CHAPTER	TITLE	PAGE NO.
01	INTRODUCTION	03
02	LITERATURE REVIEW & OBJECTIVES OF PRESENT WORK	06
03	COMPARATIVE ASSESSMENT OF DESIGN METHODOLOGIES	11
04	3-D CFD ANALYSIS OF BCRT & FCRT CENTRIFUGAL FANS	19
05	EXPERIMENTAL INVESTIGATIONS	25
06	RESULTS & DISCUSSIONS	30
07	CONCLUSIONS	42
Annexure-1	REFERENCES	-
Annexure-2	PUBLICATIONS	-

Chapter 1

INTRODUCTION

Centrifugal fans and blowers are widely used turbo machines in present industrial and domestic life. Manufacturing industries of fans and blowers seldom follows optimum design solution for individual fan/blower. Mostly their design and fabrication is based on series of successful past models or derived from fan laws and geometrical similarities.

During extensive literature review on design and performance evaluation of pumps, blowers and fans, it is observed that much research work has been carried out on local flow physics, aerodynamics and phenomena of energy transfer. It is also studied that the design methodologies suggested by different researchers differ widely. It has revealed lacuna of explicit design methodology which can give desired performance.

Based on literature review and work carried out till now, objectives of present work are compilation and experimental evaluation of existing design methodologies for radial tipped centrifugal fan, optimization of number of blades through numerical and experimental studies under varying number of blades at design as well as off-design conditions, experimental evaluation of slip factor and various losses in radial tipped centrifugal fan and presenting unified design methodology after experimental and 3-D CFD validation.

This research work is based on an industrial requirement for Fume Extraction Fan of SDS-9 texturising machine. Here variable flow is required at constant head under dust laden conditions. Radial blades are ideal for dust laden air or gas because they are less prone to blockage, dust erosion and failure. It has ideal zero slope in H-Q (head-discharge) curve to give variable discharge at constant head [1]. Hence radial blades are selected. Comparative study for forward and backward curved radial tip blade impeller fan is also planned during the course of this work.

W. J. Kearton [2] has presented his work in a paper entitled “Influence of the Number of Impeller Blades on the Pressure Generated in a Centrifugal Compressor and on its General Performance.” This is very important aspect; hence finite number of blades is to be optimized experimentally under varying speed and varying flow rate conditions. Obtained results have clearly indicated that the best performance is achieved with 16 nos. of blades in impeller. This has attributed to the fact that with 16 nos. of blades, the flow is guided with minimum separation and contributing lower frictional losses. The head coefficient, power coefficient and discharge coefficient obtained for this case are 0.00841, 2.470 and 0.289, respectively.

During further course of present work, attempts are made to design, fabricate and record performance evaluation of radial tipped centrifugal fans as per design methodology

traced out by using fundamental principles of fluid flow [1, 3, 4, 5, 6] and other design methodologies suggested by Church A. H. [7] and Osborne W. C. [8]

Each fan has been tested on experimental set up laid as per IS: 4894-1987, Indian Standard Specification for Centrifugal Fans (First Revision), Reaffirmed 1994 [9]. Observations are recorded for static pressure distribution along the flow path at variable flow rate. These are accompanied with measurements of electrical and air power parameters. Comparative study is made using forward and backward curved radial tipped vane impellers. This is done at designed and off designed rotational speeds to make stronger comparative performance assessment of suggested design methodologies. Experimental evaluation of these fans has shown performance variation at different flow sections. The performance appraisal for each design methodology concludes that design method as an individual is not performing as marked and hence there is a need to develop unified design methodology.

Based on this realization, the unified design procedure is developed which is based more on fundamental concepts and involving minimum assumptions. Best performances at different flow sections of fundamental, Church and Osborne designs are compiled together and unified design methodology is outlined. Afterward the fans are fabricated as per unified design methodology with forward and backward curved radial tipped impellers. Their performance is critically examined as per procedure on test set up described for individual design methodology. The results obtained under unified design are very much encouraging. Major performance parameters achieved are on higher side of design point during series of experiments. This shows that fan based on unified design is good enough to achieve desired performance, which not only validates the proposed unified design methodology, but also proves its strength and usefulness.

This design methodology is coded and user friendly software in visual basic for design of radial tipped centrifugal fan is developed. Impeller stress analysis is carried out using Ansys's software in order to obtain optimum vane thickness. This is done to gain advantages like reduction in rotating mass of impeller and hence saving in power consumption. This is very useful for reducing initial and long term operating cost.

The value of slip factor is essential parameter to find how much amount of energy is transferred to the fluid. It is a variable parameter and dependent of impeller geometry, flow rate, specific speed and various other factors. Experiments are made to find out variation of slip factor along blade profile at exit and normal to circumference of impeller. Special three-hole and five-hole probes are designed, developed & calibrated for measuring and sensing the local velocities. Overall value of slip factor is calculated by averaging all local values.

The experimental value of slip factor is found to be 3 to 16% less as compared to various correlations available in literature [10, 11].

In order to study flow behavior, 3-D CFD flow simulation is carried out for backward and forward curved radial tipped blade centrifugal fan using ANSYS' GAMBIT and FLUENT software. The flow simulation results again validate the design methodology and offers standard fan performance parameters. The results of CFD analysis are closer to experimental results. The careful study of flow simulation reveals the existence of recirculation of flow in the vicinity of tongue region and thus focuses towards the need of redesigning the tongue radius.

Losses proposed by various researchers differ widely. Hence, experimental investigations on losses are made to get enhanced performance of radial tipped centrifugal fan. The impeller losses are major contributor and found 68%of total hydraulic losses. This confirms the study of Andre Kovats [12] and R J Kind [13]. Volute losses are also significant and they contribute 31% of total hydraulic losses. This acknowledges the work Y.Senoo and H.Hayami [14], stating that 30% or more kinetic energy at diffuser exit remains unconverted to pressure energy. Leakage losses are found 15% of actual discharge, while mechanical losses are observed 12% of input power. More deviation of exit velocities is observed in small size impellers. It confirms that shorter blade passage height (r_2-r_1) produces more slip.

An uncertainty analysis is carried out according to Kline and Meclintock method [15], all the uncertainties in measurements are observed to be well within ± 5.0 % of design point values.

In nutshell, the numerically and experimentally validated unified design methodology, experimental evaluation of slip factor of radial tipped fans, loss analysis and the 3-D CFD analysis for flow simulation, emerges as the major outcome of this work.

Chapter 2

LITERATURE REVIEW AND OBJECTIVES OF PRESENT WORK

Spannhake's has made pioneering work in the field of turbomachines. He has defined flow physics and gave definitions of associated terminology. His work is presented in "Centrifugal pumps, turbines and propellers" (1934) [16]. His well known successor Wislieenus has extended his work and published as "Fluid Mechanics of Turbomachinery" (1945) [17].

During this period, W. J. Kearton [2] observed breaks in characteristic curves by carrying accurate tests. He had noticed that number of impeller blades has significant effect on fan performance curves.

Austin H. Church [7] has probably made the first attempt to compile the design methodology for pumps and blowers. While Eck Bruno [3], Kovats [12], William Osborne [8], Whitfield & Baines [4] had extended the work of Church and presented detailed analysis of design parameters for centrifugal, axial and cross flow fans and blowers.

Slip loss is defined as the ratio of actual & ideal values of the whirl components at exit of impeller. It has significant effect on fan performance. Stodola [1, 3] developed first useful method for slip factor approximation. He correlated slip factor and finite number of blades. Stodola claimed that average direction of discharge varies from the blade angle β_2 due to number of blades and relative circulation in vane to vane plane. This is also responsible for the reduction in output. Several co-relations as well as empirical equations are used in literature to estimate slip factor. Other slip factor correlations in literature are given by Balje, Stanitz, Bruno-Eck, [1, 3] etc. According to all these researchers, the major cause of slip factor are due to relative eddies generated in vane to vane plane. This is dependent on impeller geometry, flow rate, specific speed and various other parameters.

Overall efficiency of any turbomachine depends on shaft power input and airpower developed considering various losses occurring at different stages. Hydraulic or pressure losses, mechanical or power losses and leakage losses are major occurring losses when any fluid passes from inlet duct to outlet duct of a turbomachine. There is a basic need to understand the sources of these losses in turbo machine and consequently a mechanism to be evolved to estimate and minimize these losses accurately.

The work done by different researchers during last few decades in the field of turbomachines, centrifugal pump, centrifugal fan, blower compressor, slip factor, losses and CFD analysis are briefly summarized below:

Sr. No.	Researchers	Abstract of Work Carried Out	Year
1.	R. C. Worster [18]	Stated that at low flow rates, volute induces flow recirculation in the impeller and at high flow rates, severe energy loss occurs in the discharge line.	1963
2.	Andre Kovats [12]	Approximated Friction loss in the impeller as 50% greater than that of calculated for losses in duct and at higher speed, when Reynolds number decreases, the disc friction loss also decreases, like the hydraulic loss inside the impeller.	1964
3.	Church Austin [7]	Leakage has no effect on the head of fan/blower but it lowers the capacity and increases the power required. Between impeller and diffuser (volute) outlets, the losses will be much higher as converting kinetic energy into pressure energy is inefficient process. About 40 – 60% of velocity head will appear as pressure, remainder being lost in turbulence and friction.	1966
4.	Osborne William C. [8]	Inter blade circulation results in a reduction of the work done by the impeller. Pressure losses occurs within the fan assembly due to, a. Turning of air through 90° from axial to radial direction b. Flow separation in blade passages c. Retardation of flow velocity and eddy formation in passages of casing Power loss occurs due to fluid drag on the reverse surface of the impeller back plate.	1966
5.	Prasad, Ganeshan & Prithviraj [19]	It was found that the fluid deviation at impeller outlet increased with reduction in volume flow. The flow at the impeller outlet became more and non-uniform at low volumes and the deviation in meridional plane was found to be more near the back shroud.	1973
6.	Yedidiah Sh. [10]	Presented a new model of slip factor that resolves basic discrepancies observed between old theories. He presented factual evidences which proved that the slip factor for a given impeller is not constant, but varies with	1974

		the flow rate.	
7.	Robert Kazar & John Lynch [20]	Presented CF fan design for energy conservation with balanced economic considerations.	1978
8.	S. Sundaram [21]	The optimization of number of blades of centrifugal fan impeller involves a maximization problem of multivariable function with fluid dynamic constraints. Experimental data based on a simple variation in blade number alone, keeping other parameters constant, will not yield optimum blade numbers for a global maximum hydraulic efficiency.	1978
9.	Patel, Patel & Shah [22]	Presented the method, which gives the range of design constants and rapid selection for optimum design.	1981
10.	Lal & Vasandani [23]	Studied slip factor effect on designing of impeller and concluded that slip factor reduces due to non-uniform velocity distribution at impeller exit.	1988
11.	Y.Senoo and H.Hayami [14]	30% or more of the kinetic energy at the exit of the impeller is not converted into pressure in the diffuser. Such pressure loss reduces hydraulic efficiency.	1989
12.	R. Ajithkumar [11]	Slip factor is a function of number of vanes, diameter ratio, outlet blade angle and flow conditions after impeller. When blade angle is smaller, frictional losses are larger.	1990
13.	Mishra Bela [24]	Critically studied the design methodology as suggested by Dr. Ing Bruno Eck.	1997
14.	R. J. Kind [13]	Describes a method for predicting flow behavior and performance for centrifugal fans of the squirrel-cage type. Studied that volute can have a strong influence on performance characteristics of the machine. The inlet losses and volute friction losses are relatively unimportant. Blading losses are, however, very important and are responsible for approximately half of the overall losses. The other major loss is 'dump or sudden enlargement loss. The flow rate through the gap has a substantial effect on the dump loss because higher gap flow rates mean higher velocities in the volute and thus lower dump losses.	1997
15.	Kwang-Yong Kim, Seoung-Jin	Described the numerical analysis method by Reynolds-averaged Navier-Stokes equations taking standard k-ε	2004

	Seo [25]	turbulence model and discretized with finite volume approximations.	
16.	T. Meakhail, S. Park [26]	Studied Impeller-Diffuser-Volute Interaction in a Centrifugal Fan with numerical flow simulation.	2005
17.	ANSYS Inc. [27]	With the help of given tutorial, it illustrates the procedure for setting up and solving a problem using MRF approach specifying different frames of reference for different fluid zones and setting relative velocity of each wall using the segregated solver.	2006
18.	Hong Yang, Dirk Nuernberger, Hans-Peter Kersken [28]	Developed a three-dimensional hybrid structured-unstructured grid Reynolds-averaged Navier-Stokes (RANS) solver to simulate flows in complex turbomachinery geometries.	2006
19.	Mohamad Memardezfouli, Ahmad Nourbakhsh [29]	Experimental slip factors are compared with the calculated theoretical values and found that they are in good agreement at design point conditions but deviates at off design conditions.	2009

Literature Review Conclusions:

The survey of literature indicates that specific and focused research work has been carried out all over the world on local flow physics, aerodynamics and phenomena of energy transfer. It includes study of various parameters affecting the power, head and efficiencies. But major lacuna exists towards the availability of following:

1. An explicit centrifugal fan design procedure, validated through experiments.
2. Optimum number of blades, which can offer design point performance with maximum efficiency.
3. Comparative evaluation of empirical and experimental slip factor.

Objectives of Present Work:

Looking to literature review cited above, and works carried out till now, objectives of present work are set as following:

1. Compilation and experimental evaluation of existing design methodologies for radial tipped centrifugal fan.

2. Optimization of number of blades through numerical and experimental studies under varying number of blades at design as well as off-design conditions.

3. Developing unified design methodology for radial tipped centrifugal fan and checking its validity through experiments.

4. Flow simulation through 3-D CFD approach and numerical validation of unified design methodology for radial tipped centrifugal fan.

5. Experimental evaluation of slip factor and various other losses occurring in radial tipped centrifugal fan.

Chapter 3

COMPARATIVE ASSESSMENT OF DESIGN METHODOLOGIES

Three systematic design methodologies for centrifugal fan are traced out after comprehensive literature review. Focusing on these three design methods, comparative assessment is made mathematically and then experiments are carried out to get optimum design solution. These design methodologies are summarized as under.

3.1.1 Design Procedure Laid Down using Fundamental Principles [1, 3, 4, 5, 6]

This design procedure is based on fundamental principles of fluid flow with continuity equations. Energy balance is established at fan inlet, intermediate stage of impeller and outlet stage of volute/scroll casing. During this process stage velocity, pressure and discharge at different stages are calculated. Flat front and back shrouds are selected for ease of impeller fabrication.

3.1.2 Design Procedure as Suggested by Austin Church [1, 7, 8]

Austin Church has done pioneering work to establish design methodology for pumps and blowers. He has presented his design with stage compressibility effect. He has also considered density changes at various flow sections with respect to change in temperature and pressure. Thus volume flow rate gets changed continuously. The dimensions of the air passage are calculated in accordance to this variation in volume flow. Stage pressure ratio between atmosphere to inlet eye, inlet eye to impeller inlet, impeller inlet to impeller outlet and impeller outlet to casing outlet are calculated individually. Church has used empirical impeller pressure coefficient K'. This K' lies in the range of 0.5-0.65.

3.1.3 Design Procedure as Suggested by William .C. Osborne [1, 8]

William. C. Osborne has made very good attempt to use simple flow physics to design fans/blowers. He has used empirical relations for eye velocity, meridian velocity and casing velocity with respect to impeller tip peripheral velocity.

Relative velocity is considered same for inlet and outlet conditions. This is one of the major limitations of this design. Suction, impeller, volute pressure losses and leakage losses are calculated separately.

3.2 Comparative Assessment of Designs

An industrial requirement for fume extraction fan of SDS-9 texturising machine is taken as base data for research work. In this case, variable flow is required at constant head under dust laden conditions. Radial blades are ideal for dust laden air or gas because they are less prone to blockage, dust erosion and failure. It has ideal zero slope in H-Q (head-discharge) curve to give variable discharge at constant head [1]. Hence radial blades are selected. Summary of input parameters is listed below:

Discharge Q = 0.5 m^3/s

Static Suction Pressure = -196.4 N/m^2

Static Delivery Pressure = 784.8 N/m^2

Static Pressure Gradient ΔPs= 981.2 Pa

Speed of impeller rotation N = 2800 rpm

Air Density = 1.165 kg/m^3

Number of blades z= 16 [30]

Outlet Blade Angle $\beta_2 = 90°$

Suction Temperature Ts=30 °C = 303° K

Atmospheric Pressure $P_{atm}=1.01325 \times 10^5$ Pa

Atmospheric Temperature $T_{atm}=30°$ C = 303° K

Geometrical and other parameters as obtained by fundamental [1, 3, 4, 5, 6], Church [7] and Osborne [8] design methodologies are given in Table 3.1. This is for identical input data. Iterations are made to get optimum design parameters. This is done by taking hydraulic, mechanical and capacity losses into consideration. It is clearly observed by comparative assessment that there is wide variation in most of the parameters, achieved under these design methodologies.

Table 3.1 Comparative Assessment of Design Methodologies

Parameters		**Unit**	**FUNDAMENTAL**	**CHURCH**	**OSBORNE**
	At Impeller Outlet				
Peripheral Velocity	U_2	m/s	37.64	43.025	43.36
Relative Velocity	W_2	m/s	12.61	16.15	8.672
Meridian Velocity	Vm_2	m/s	12.61	16.15	8.672
Absolute Velocity	V_2	m/s	32.64	38.26	35.76
Impeller Diameter	d_2	mm	257	295	296

Width Of Blade	b_2	mm	53.2	34.85	65.2
Air Angle	α_2	Deg.	22.72°	25.93°	14.03°
Blade Angle	β_2	Deg.	90°	90°	90°
	At Impeller Inlet				
Eye Diameter	d_o	mm	174	188	191.6
Eye Velocity	V_{eye}	m/s	21.04	18	17.344
Peripheral Velocity	U_1	m/s	23.152	27.841	28.72
Relative Velocity	W_1	m/s	31.29	33.79	30
Meridian Velocity	Vm_1	m/s	21.047	19	8.672
Absolute Velocity	V_1	m/s	21.047	19	8.672
Impeller Diameter	d_1	mm	158	191	196
Width Of Blade	b_1	mm	53.2	52	98.4
Air Angle	α_1	Deg.	90°	90°	90°
Blade Angle	β_1	Deg.	42.27°	35°	16.80°
Leakage Loss	Q_L	m^3/s	0.020	0.0148	0.02481
Pressure Losses In Impeller	dp_{im}	Pa	218.39	155.36	199.48
	Volute Casing				
Width Of Casing	B_v	mm	112	104.55	130.4
Outlet Velocity Of Casing	V_3	m/s	32.012	19.96	31.95
Diameter Of Casing at 0°	d_3	mm	267	305	306
Diameter Of Casing at 360°	d_4	mm	546	774	542
Volute Tongue Angle	θ_t	Deg.	9.89°	11.24°	16.58°
Radius of Tongue	R_t	mm	138	161.80	159
	Losses, Power and Efficiencies				
Casing Pressure Losses		Pa	0.092	171.28	3.38
Disk Friction Torque	T_d	Nm	0.011	0.0059	0.0223
Power Loss Disk Friction	P_{df}	Watts	3.22	1.73	6.534
Power Required To Run Fan	P	Watts	793.68	928.39	768.23
Hydraulic Efficiency	η_{hy}	%	81.8%	75%	84.24%
Volumetric Efficiency	η_{vol}	%	96.2%	97%	95.27%
Total Efficiency	η_T	%	78.6%	72.75%	80.25%
Shaft Diameter	d_s	mm	11.71	12.35	11.59
Blade Profile Radius	R_b	mm	87.8	81.03	65.5

3.3 Fabrication and Testing

Radial tipped centrifugal fans as suggested by above different design methodologies are fabricated as per parameters given in Table 3.1. These fans are then tested as per test standard IS: 4894-1987, Indian Standard Specification for Centrifugal Fans, 1994 [9]. Precise and calibrated instruments are used to get consistent results.

Experimental set up, procedure and results are given in detail in subsequent chapters. Based on experimental results obtained (refer chapter 6.3.4), it is observed that there exists a wide performance difference amongst fans under study. All fans are not performing as per mark. It has revealed that there is a need to develop unified design methodology.

3.4 Unified Design Methodology

Successful outcomes of fundamental [1, 3, 4, 5, 6], Church [7] and Osborne [8] designs are incorporated together and a new design methodology for radial tipped centrifugal fan is developed. This is named as unified design methodology for radial tipped centrifugal fan.

Table 3.2 represents various parameters obtained under unified design methodology. Two successive iterations are made to include effect of major possible volumetric and pressure losses occurring at inlet, impeller and volute sections losses. This is made to get optimum parameters for fabrication.

Table 3.2 Various Parameters as Obtained by Unified Design Methodology

Parameters		**Unit**	**0^{th} iteration**	**1^{st} iteration**	***2^{nd} iteration**
	At Impeller Outlet				
Peripheral Velocity	U_2	m/s	37.37	42.79	43.67
Relative Velocity	W_2	m/s	12.70	11.33	11.052
Meridian Velocity	Vm_2	m/s	12.70	11.33	11.052
Absolute Velocity	V_2	m/s	32.61	36.21	36.80
Impeller Diameter	d_2	mm	255	292	* 298
Width Of Blade	b_2	mm	51.1	51.76	52
Air Angle	α_2	Deg.	22.93°	18.23°	17.48°
Blade Angle	β_2	Deg.	90°	90°	90°
	At Impeller Inlet				
Eye Diameter	d_o	mm	174	176	176

Eye Velocity	V_{eye}	m/s	21	21.33	21.36
Peripheral Velocity	U_1	m/s	23.152	23.44	23.44
Relative Velocity	W_1	m/s	31.29	31.68	31.68
Meridian Velocity	Vm_1	m/s	21.047	21.31	21.31
Absolute Velocity	V_1	m/s	21.047	21.31	21.31
Impeller Diameter	d_1	mm	158	160	160
Width Of Blade	b_1	mm	51.1	51.76	52
Air Angle	α_1	Deg.	90°	90°	90°
Blade Angle	β_1	Deg.	42.27°	42.27°	42.27°
Leakage Loss	Q_l	m^3/s	0.0199	0.2037	0.02038
Pressure Losses In Impeller	dp_{im}	Pa	154.54	163.91	165.013
	Volute Casing & Others				
Width Of Casing	b_v	mm	109.35	110.76	111.28
Outlet Velocity Of Casing	V_3	m/s	23.80	27.23	27.86
Scroll Radius	r_3	mm	181.56	207.904	212.176
Scroll Radius	r_4	mm	213.435	244.404	249.426
Scroll Radius	r_5	mm	245.31	280.904	286.676
Scroll Height	Hs	mm	285.6	327.04	333.76
Radius of tongue	R_t	mm	25.5	29.2	29.8
Casing Pressure Losses	D_p	Pa	150.45	192.39	199.98
Disk Friction		Nm	0.01062	0.0209	0.02314
Power Loss In Disk Friction	P_{df}	Watts	3.11	6.12	6.78
Hydraulic Efficiency	η_{hy}	%	76.29%	73.36%	72.89%
Volumetric Efficiency	η_{vol}	%	96.17%	96.08%	96.08%
Total Efficiency	η_t	%	73.37%	70.49%	70.03%
Power Required To Run Fan	P	Watts	914.51	993.48	1005.62
Shaft Diameter	d_s	mm	12.28	12.63	12.68

*After 2nd iteration, there is a little change in impeller outlet diameter d_2 and hence no further iterations are required.

3.5 Designing Software

Design calculations and iterations is cumbersome process. Number of variables and coefficients should be taken care of while designing. It needs number of iterations before reaching to optimum parameters. In present era of computerization, it can be made faster by writing sequential program. User-friendly software in Visual basic is prepared for designing this radial tipped centrifugal blower/fan based on proposed unified design methodology. Few frames from this software are presented in Fig. 3.1 to 3.6.

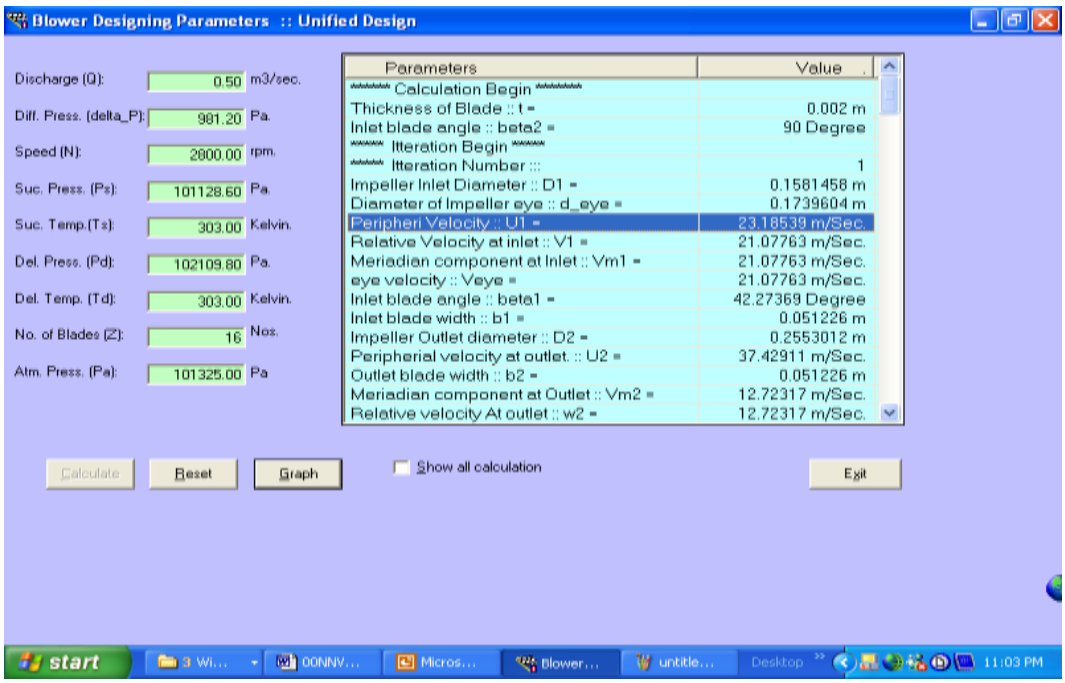
Blower Designing Parameters :: Unified Design
Discharge (Q): 0.50 m3/sec.
Diff. Press. (delta_P): 981.20 Pa.
Speed (N): 2800.00 rpm.
Suc. Press. (Ps): 101128.60 Pa.
Suc. Temp.(Ts): 303.00 Kelvin.
Del. Press. (Pd): 102109.80 Pa.
Del. Temp. (Td): 303.00 Kelvin.
No. of Blades (Z): 16 Nos.
Atm. Press. (Pa): 101325.00 Pa
Parameters
Value
****** Calculation Begin ******
Thickness of Blade :: t = 0.002 m
Inlet blade angle :: beta2 = 90 Degree
****** Itteration Begin ******
****** Itteration Number ::: 1
Impeller Inlet Diameter :: D1 = 0.1581458 m
Diameter of Impeller eye :: d_eye = 0.1739604 m
Peripheri Velocity :: U1 = 23.18539 m/Sec.
Relative Velocity at inlet :: V1 = 21.07763 m/Sec.
Meriadian component at Inlet :: Vm1 = 21.07763 m/Sec.
eye velocity :: Veye = 21.07763 m/Sec.
Inlet blade angle :: beta1 = 42.27369 Degree
Inlet blade width :: b1 = 0.051226 m
Impeller Outlet diameter :: D2 = 0.2553012 m
Peripherial velocity at outlet. :: U2 = 37.42911 m/Sec.
Outlet blade width :: b2 = 0.051226 m
Meriadian component at Outlet :: Vm2 = 12.72317 m/Sec.
Relative velocity At outlet :: w2 = 12.72317 m/Sec.
Calculate
Reset
Graph
Show all calculation
Exit
start
11:03 PM

Fig. 3.1 Input Data Entry Screen with Start of Calculations

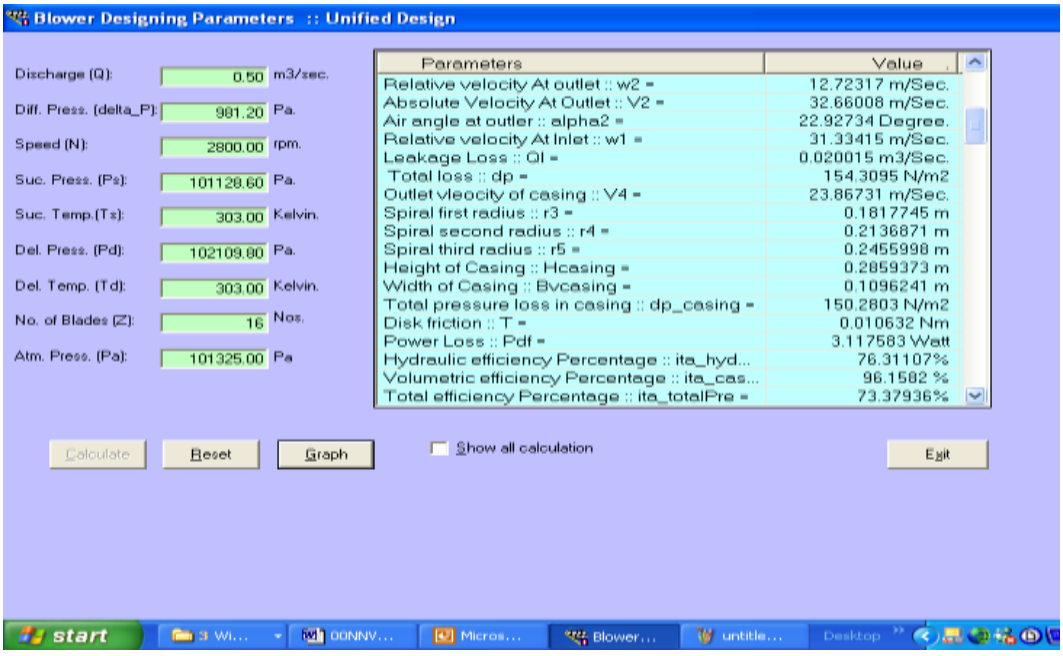
Blower Designing Parameters :: Unified Design
Discharge (Q): 0.50 m3/sec.
Diff. Press. (delta_P): 981.20 Pa.
Speed (N): 2800.00 rpm.
Suc. Press. (Ps): 101128.60 Pa.
Suc. Temp.(Ts): 303.00 Kelvin.
Del. Press. (Pd): 102109.80 Pa.
Del. Temp. (Td): 303.00 Kelvin.
No. of Blades (Z): 16 Nos.
Atm. Press. (Pa): 101325.00 Pa
Parameters
Value
Relative velocity At outlet :: w2 = 12.72317 m/Sec.
Absolute Velocity At Outlet :: V2 = 32.66008 m/Sec.
Air angle at outler :: alpha2 = 22.92734 Degree.
Relative velocity At Inlet :: w1 = 31.33415 m/Sec.
Leakage Loss :: Ql = 0.020015 m3/Sec.
Total loss :: dp = 154.3095 N/m2
Outlet vleocity of casing :: V4 = 23.86731 m/Sec.
Spiral first radius :: r3 = 0.1817745 m
Spiral second radius :: r4 = 0.2136871 m
Spiral third radius :: r5 = 0.2455998 m
Height of Casing :: Hcasing = 0.2859373 m
Width of Casing :: Bvcasing = 0.1096241 m
Total pressure loss in casing :: dp_casing = 150.2803 N/m2
Disk friction :: T = 0.010632 Nm
Power Loss :: Pdf = 3.117583 Watt
Hydraulic efficiency Percentage :: ita_hyd... 76.31107%
Volumetric efficiency Percentage :: ita_cas... 96.1582 %
Total efficiency Percentage :: ita_totalPre = 73.37936%
Calculate
Reset
Graph
Show all calculation
Exit
start

Fig. 3.2 Output Parameters

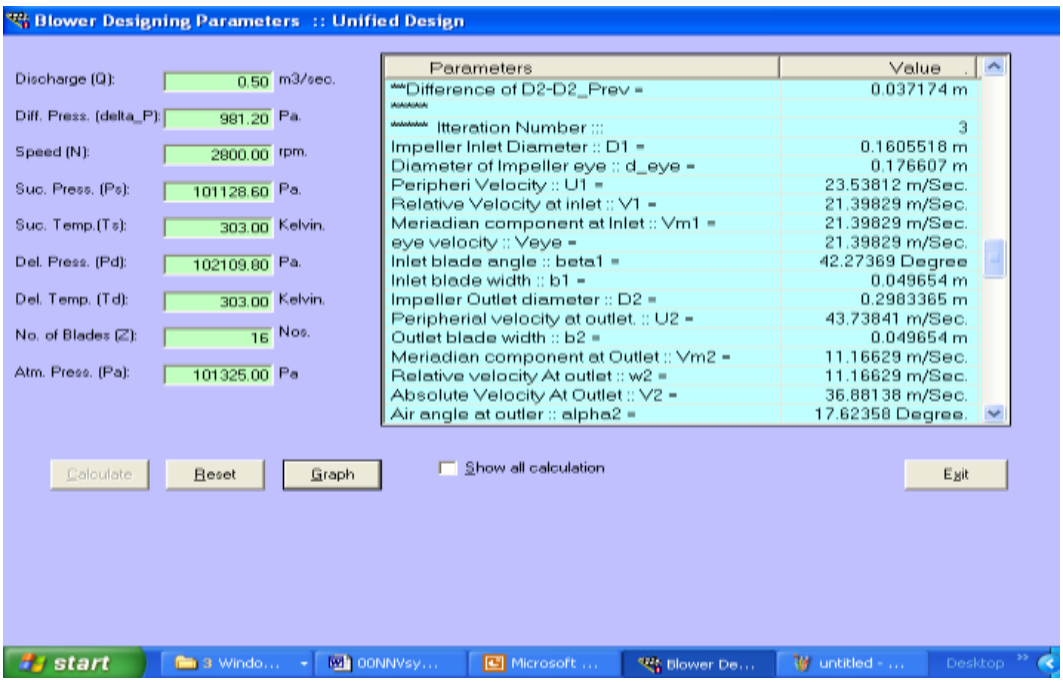
Blower Designing Parameters :: Unified Design
Discharge (Q): 0.50 m3/sec.
Diff. Press. (delta_P): 981.20 Pa.
Speed (N): 2800.00 rpm.
Suc. Press. (Ps): 101128.60 Pa.
Suc. Temp.(Ts): 303.00 Kelvin.
Del. Press. (Pd): 102109.80 Pa.
Del. Temp. (Td): 303.00 Kelvin.
No. of Blades (Z): 16 Nos.
Atm. Press. (Pa): 101325.00 Pa
Parameters
Value
Difference of D2-D2_Prev = 0.037174 m
Itteration Number ::: 3
Impeller Inlet Diameter :: D1 = 0.1605518 m
Diameter of Impeller eye :: d_eye = 0.176607 m
Peripheri Velocity :: U1 = 23.53812 m/Sec.
Relative Velocity at inlet :: V1 = 21.39829 m/Sec.
Meriadian component at Inlet :: Vm1 = 21.39829 m/Sec.
eye velocity :: Veye = 21.39829 m/Sec.
Inlet blade angle :: beta1 = 42.27369 Degree
Inlet blade width :: b1 = 0.049654 m
Impeller Outlet diameter :: D2 = 0.2983365 m
Peripherial velocity at outlet. :: U2 = 43.73841 m/Sec.
Outlet blade width :: b2 = 0.049654 m
Meriadian component at Outlet :: Vm2 = 11.16629 m/Sec.
Relative velocity At outlet :: w2 = 11.16629 m/Sec.
Absolute Velocity At Outlet :: V2 = 36.88138 m/Sec.
Air angle at outler :: alpha2 = 17.62358 Degree.
Calculate
Reset
Graph
Show all calculation
Exit
start

Fig. 3.3 Output Parameters at 3rd Iteration

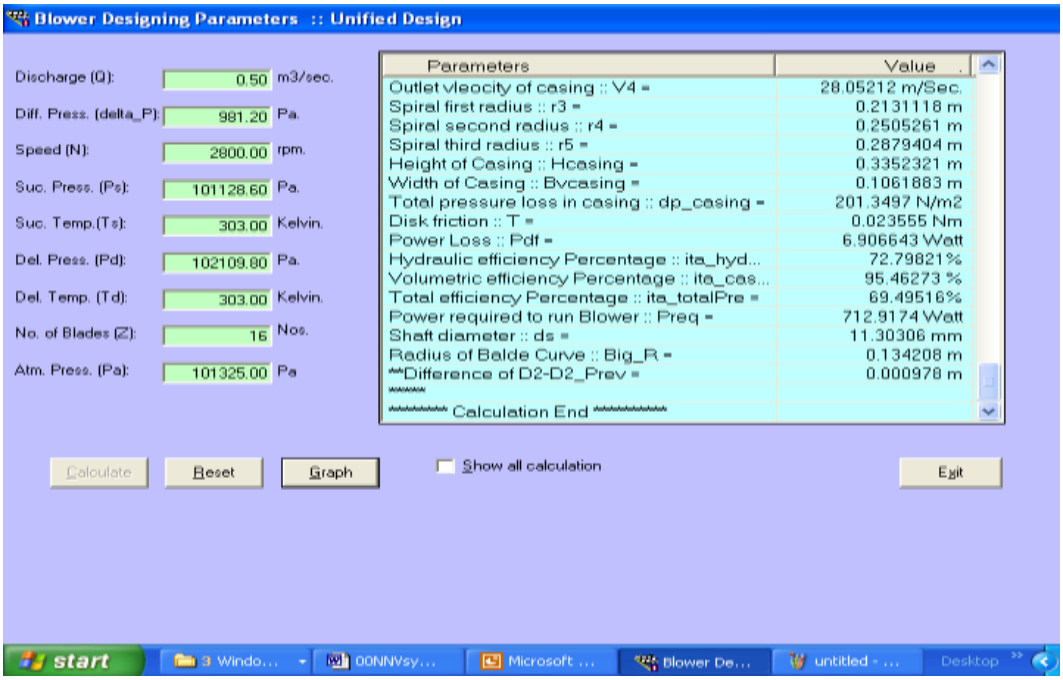
Blower Designing Parameters :: Unified Design
Discharge (Q): 0.50 m3/sec.
Diff. Press. (delta_P): 981.20 Pa.
Speed (N): 2800.00 rpm.
Suc. Press. (Ps): 101128.60 Pa.
Suc. Temp.(Ts): 303.00 Kelvin.
Del. Press. (Pd): 102109.80 Pa.
Del. Temp. (Td): 303.00 Kelvin.
No. of Blades (Z): 16 Nos.
Atm. Press. (Pa): 101325.00 Pa
Parameters
Value
Outlet vleocity of casing :: V4 = 28.05212 m/Sec.
Spiral first radius :: r3 = 0.2131118 m
Spiral second radius :: r4 = 0.2505261 m
Spiral third radius :: r5 = 0.2879404 m
Height of Casing :: Hcasing = 0.3352321 m
Width of Casing :: Bvcasing = 0.1061883 m
Total pressure loss in casing :: dp_casing = 201.3497 N/m2
Disk friction :: T = 0.023555 Nm
Power Loss :: Pdf = 6.906643 Watt
Hydraulic efficiency Percentage :: ita_hyd... 72.79821%
Volumetric efficiency Percentage :: ita_cas... 95.46273 %
Total efficiency Percentage :: ita_totalPre = 69.49516%
Power required to run Blower :: Preq = 712.9174 Watt
Shaft diameter :: ds = 11.30306 mm
Radius of Balde Curve :: Big_R = 0.134208 m
Difference of D2-D2_Prev = 0.000978 m
Calculation End
Calculate
Reset
Graph
Show all calculation
Exit
start

Fig. 3.4 Output Parameters at End of Calculations

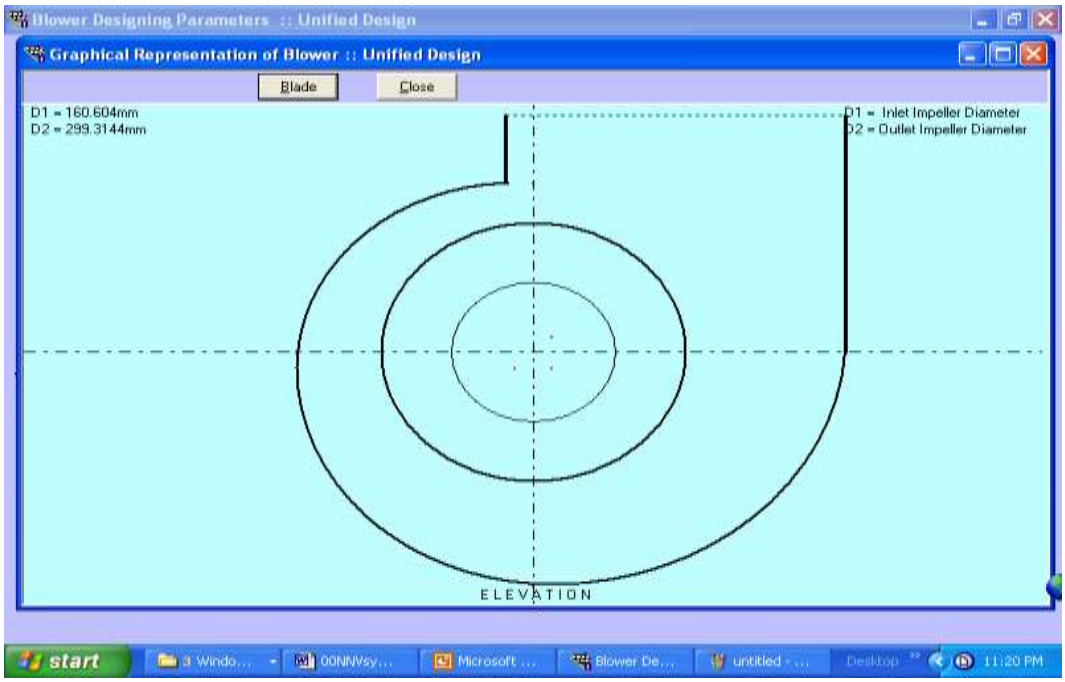

Fig. 3.5 Volute Casing Design and Fabrication Drawing

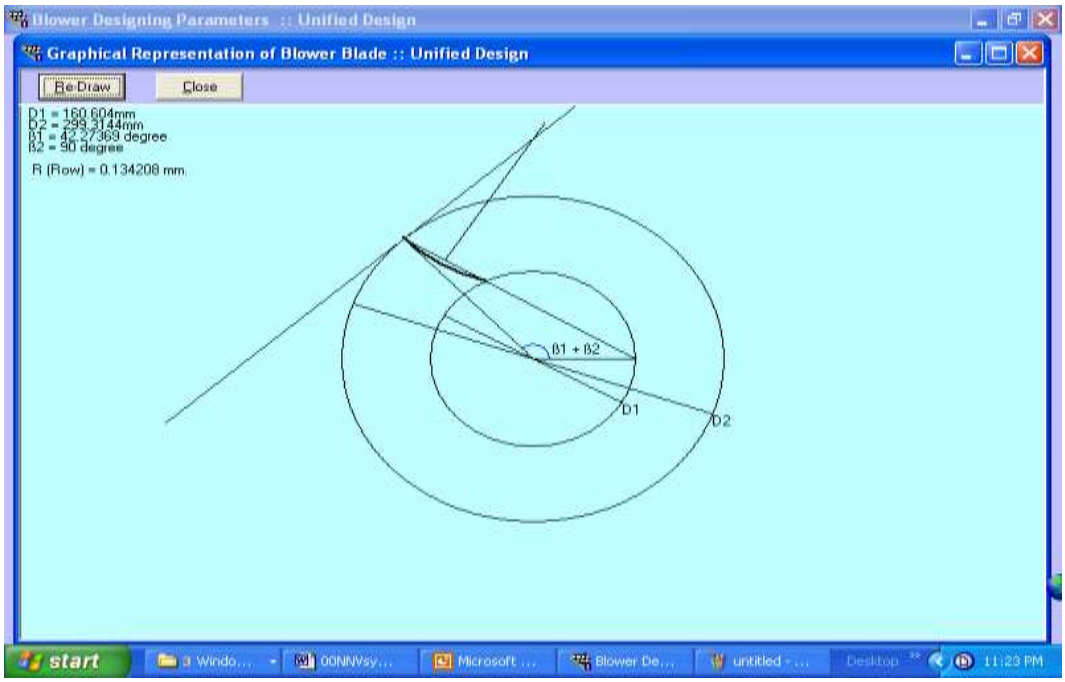

Fig. 3.6 Blade Profile Design

Chapter 4

3-D CFD ANALYSIS OF BCRT AND FCRT CENTRIFUGAL FANS

Backward and forward curved radial tipped blade centrifugal fans designed on unified method are simulated using three dimensional computational fluid dynamic (CFD) approach. Numerical analysis is carried out to study and visualize flow characteristics at design and off-design conditions. Initializing conditions are given by varying mass flow rate, rotational speed and number of blades.

The 3-D CFD analysis is carried out by using ANSYS' GAMBIT and FLUENT software using 'Reynolds-averaged Navier-Stokes' equations (RANS) and 'Realizable k-ε model'. 'Standard' wall function is used to resolve wall flows and 'Simple' algorithm is used for coupling pressure and velocity [25, 26, 27].

Steady flow and very low pressure ratio conditions are prevailing in this case. Hence, moving reference frame (MRF) approach [27] is used to impose rotational field to impeller zone of centrifugal fan. This condition is achieved by using moving wall concept keeping junction at zero relative rotational speed to adjacent cell zone. Shadow wall method is used at interior surface to create continuous flow path between moving and stationary zones.

Mass flow rate at nozzle inlet is used as inlet boundary condition. Zero gradient outflow condition is used at casing outlet. This is done for fully developed flow conditions. At inlet boundary condition, 5% turbulent intensity and 0.5 turbulent length scale is applied [27]. This is calculated based on cube root of domain volume and used for turbulence specifications. 'No-slip' boundary condition is used for all walls. The discharge at nozzle inlet at each step of rotational speed of impeller is varied by varying inlet mass flow rate.

The discretization of pressure is done by using 'PRESTO' scheme. This scheme is generally used to obtain good results if there is vortex flow field. The momentum, turbulent kinetic energy, turbulent dissipation rate and energy are discretized by using 'second order upwind method'. Maximum residuals are less than 10^{-5} [27, 28, 31] as solution convergence criterion.

The results of this converged simulation have given good insight on flow behavior occurring inside the centrifugal fan.

Figure 4.1 shows consolidated algorithm for overall work carried out during the course of this project work.

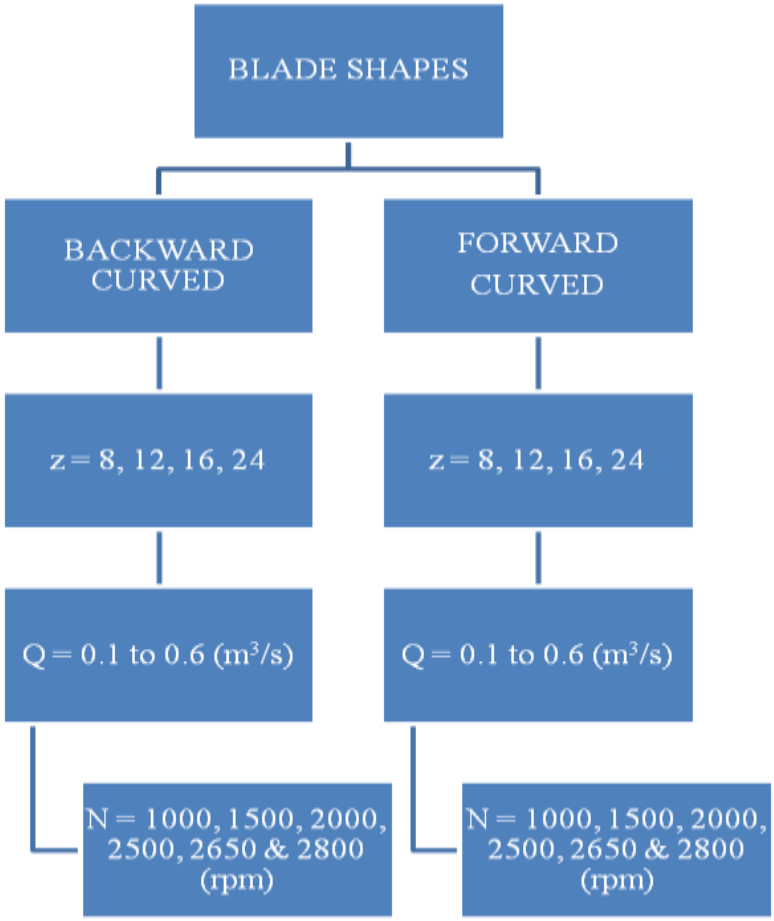

Figure 4.1 Consolidated Algorithms for Simulated Cases

The pre-processing work is done by GAMBIT (Geometry and Mesh Building Intelligent Toolkit) software. GAMBIT is very robust tool for geometry generation and it gives total user control while doing meshing work. Meshing of all zones is done by unstructured tetrahedral mesh elements [27, 28]. Impeller region has fine mesh elements as compared to inlet nozzle and volute casing. Before starting simulation process, number of elements is optimized and grid independency test is carried out.

ANSYS FLUENT software is used for post processing work. It contains broad physical modeling capabilities needed to model for flow, turbulence, heat transfer etc. This is very essential for turbomachinery simulation. It provides multiple choices in solver option. It combines gives optimum solution efficiency and accuracy for a wide range of speed regimes.

Efficient energy transfer in a centrifugal fan depends upon good impeller inlet conditions, proper blade profile, gradual change in volute casing area and overall smooth surface finish. For such energy transfer, flow lines must be parallel to each other and should generate streamlined flow within three dimensional guided passages [5, 18, 27].

Fig. 4.2 shows mesh diagram with 419633 mesh elements for volume discretization and grid independency test [27].

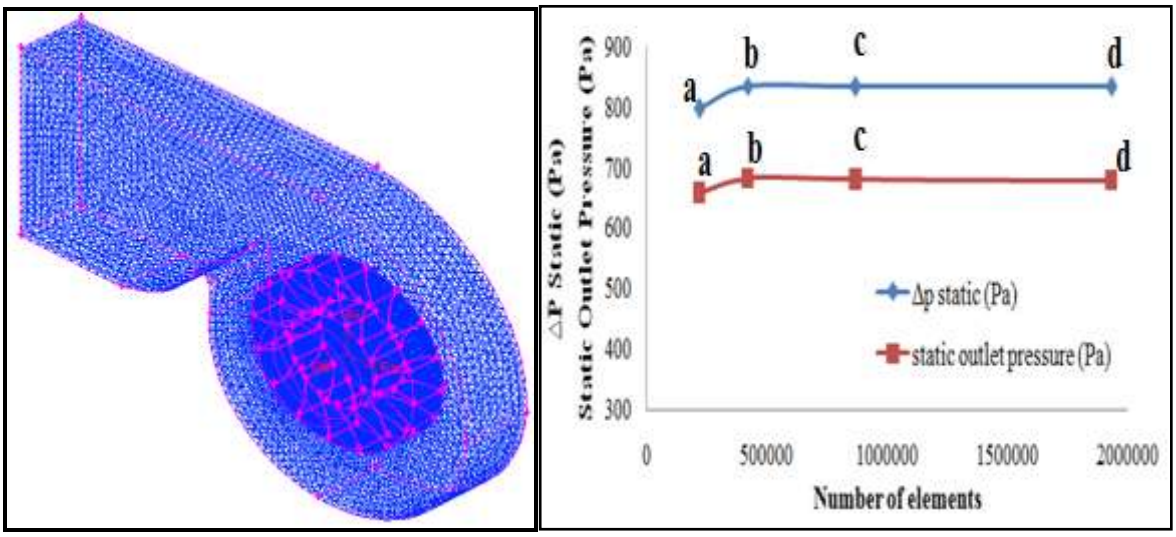

Figure 4.2 Mesh diagram and grid independency test of centrifugal fan

Figure 4.3 shows flow streamlines at mid plane cross-section for 16 numbers of blades. Here streamlines are seen parallel and efficiently guided within entire flow passage. It confirms that flow across the stage is well guided and flow leaves impeller smoothly and enters in volute casing without circulation. Volute casing progressively guides flow to outlet section of the centrifugal fan. Little flow re-circulation is observed near tongue region and a small vortex is generated when it leaves tongue region.

Similar kind of streamlined flow is observed at various axial cross sections under varying number of blades and mass flow rate conditions.

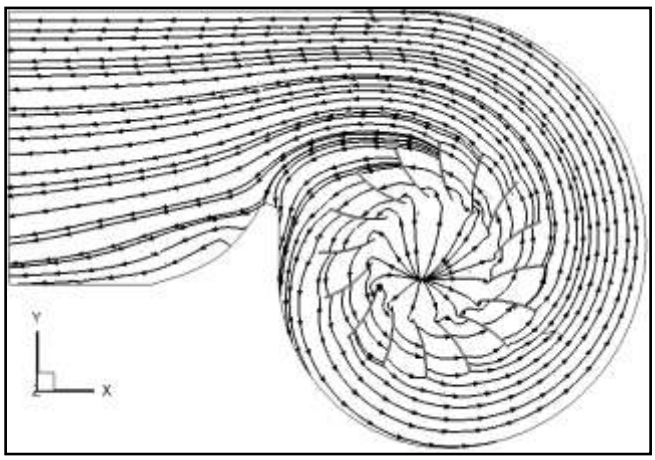

Figure 4.3 Streamlines at mid-plane of centrifugal fan for 16 nos. of blades [18, 31]

Post-processing simulation results presents flow visualization within centrifugal fan. It can be represented in the form of contours for different parameters. Various contour plots

are obtained in each case of rotational speed in the step of N = 1000, 1500, 2000, 2500, 2650 and 2800 rpm, for each set of number of blades (Z = 12, 16 and 24). Here discharge is kept constant at design value Q=0.5 m^3/s. Every case legend is plotted with identical maximum and minimum values. This can give visual comparison at a glance.

Figures 4.4 (a) to 4.4 (f) shows 3 dimensional and mid-plane profile of static pressure, total pressure and velocity magnitude at different planes along x and z axis, respectively **for backward curved radial tipped centrifugal fan** having Z=16, N=2800 rpm and Q=0.5 m^3/s.

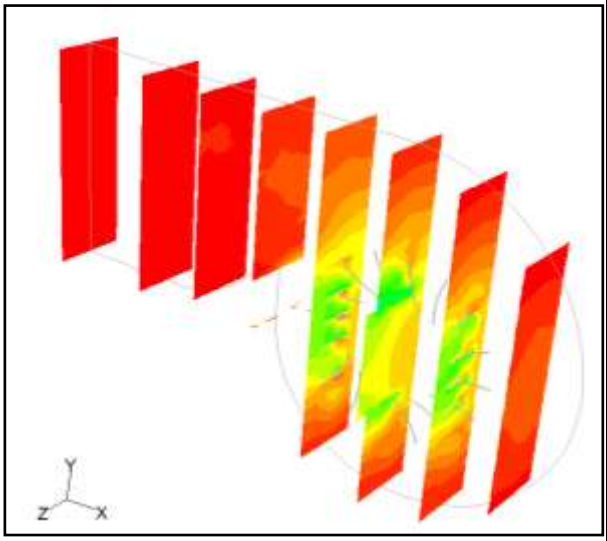

Figure 4.4 (a) Static pressure contour along 3-D planes

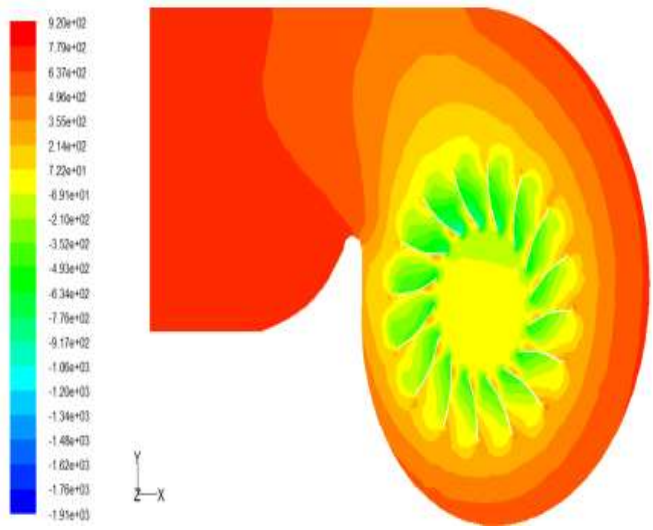

Figure 4.4 (b) Static pressure (in Pa) contour at mid-plane

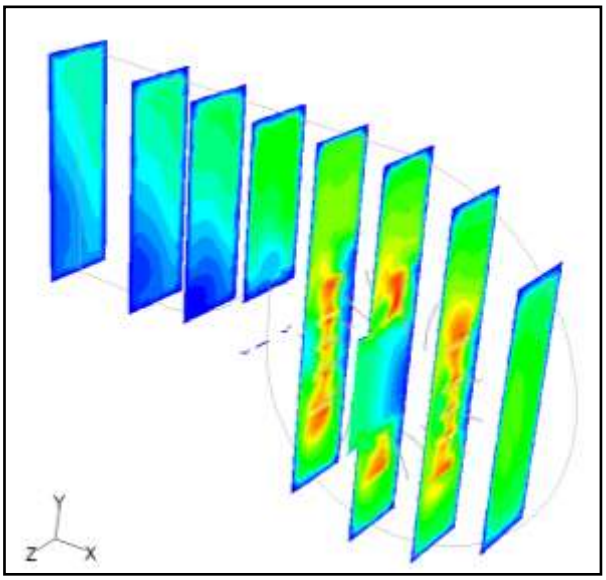

Figure 4.4 (c) Velocity magnitude Contour along 3-D planes

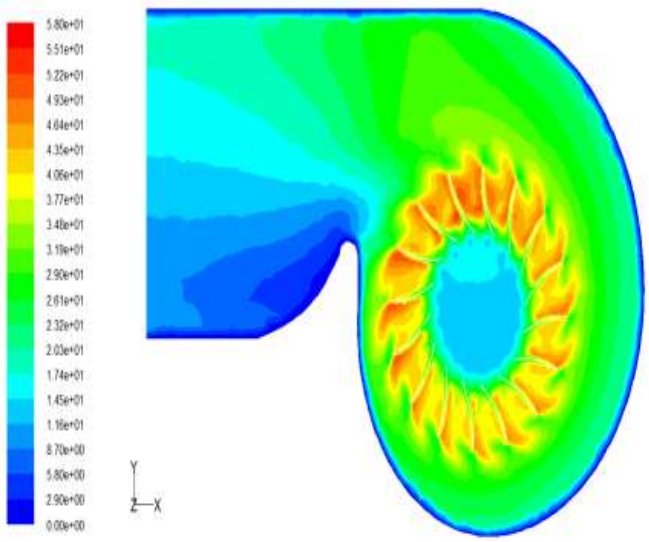

Figure 4.4 (d) Velocity magnitude (in m/s) Contour at mid-plane

From Fig. 4.4 (a) and (b) shows static pressure rise from inlet eye to fan exit. It is observed that average impeller inlet static pressure is – 146 Pa and average impeller exit static pressure is 41 Pa. It makes 187 Pa impeller stage static pressure rise. While in volute casing, static pressure increases from 41 Pa to 682 Pa. At nose radius plane, static pressure is 534 Pa. These results follow standard fan performance H-Q curves as given in literature. [8]

Fig. 4.4 (c) and (d) show velocity magnitude distribution contours. It is observed that average impeller inlet velocity is 20 m/s and average impeller exit velocity is 37 m/s. It develops 580 Pa velocity head within impeller stage. While in volute casing, inlet velocity is 37 m/s and exit velocity is 11.32 m/s. It shows that 711 Pa velocity head is recovered within vane less volute casing. Velocity distribution in exit section is distributed in to two zones. Fluid passing at nose radius is obstructed and sudden fall in velocity is observed. This reduces effective fan exit area.

Fig. 4.4 (e) and (f) shows total pressure rise from inlet eye to fan exit. It is observed that average impeller inlet total pressure is 99 Pa and average impeller exit total pressure is 861 Pa. It makes 762 Pa impeller stage stagnation pressure rise. While in volute casing, total pressure rise is from 861 Pa to 790 Pa. At nose radius plane total pressure is 800 Pa at 20.56 m/s fluid velocity. Drop in total pressure indicates substantial volute losses and flow recirculation. Static pressure rise across the fan stage is observed. These results again follow standard fan performance curves as given in literature. [8, 32]

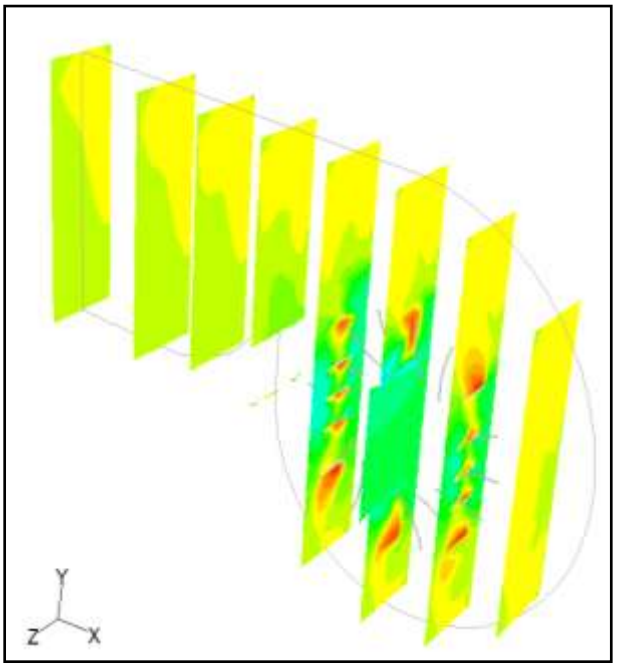

Figure 4.4 (e) Total pressure contour along 3-D planes

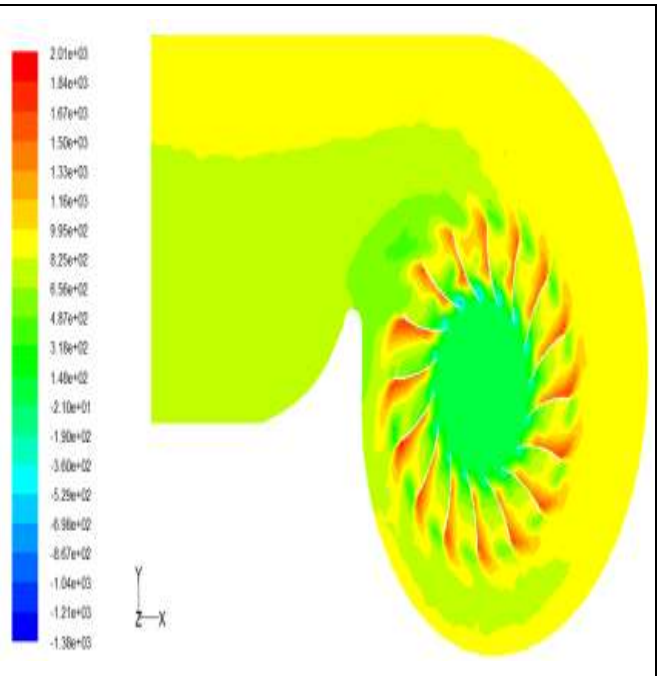

Figure 4.4 (f) Total pressure contour (in Pa) at mid-plane

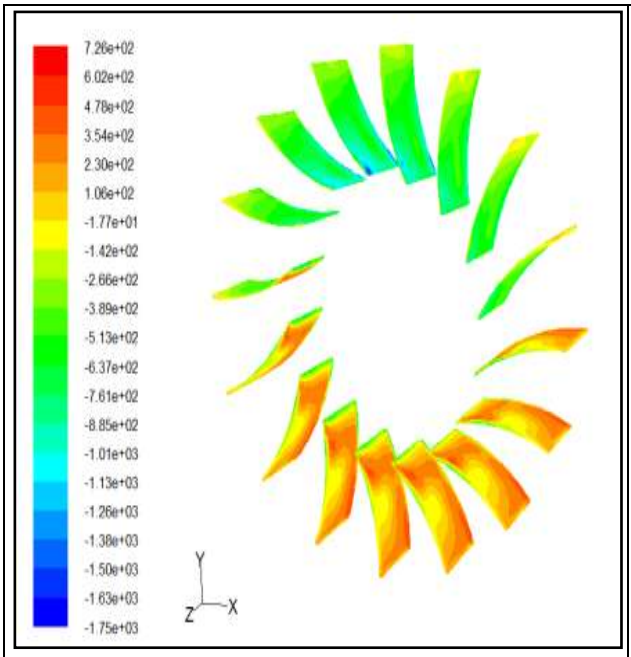

Figure 4.4 (g) shows static pressure (in Pa) blade loading over entire blade area of impeller

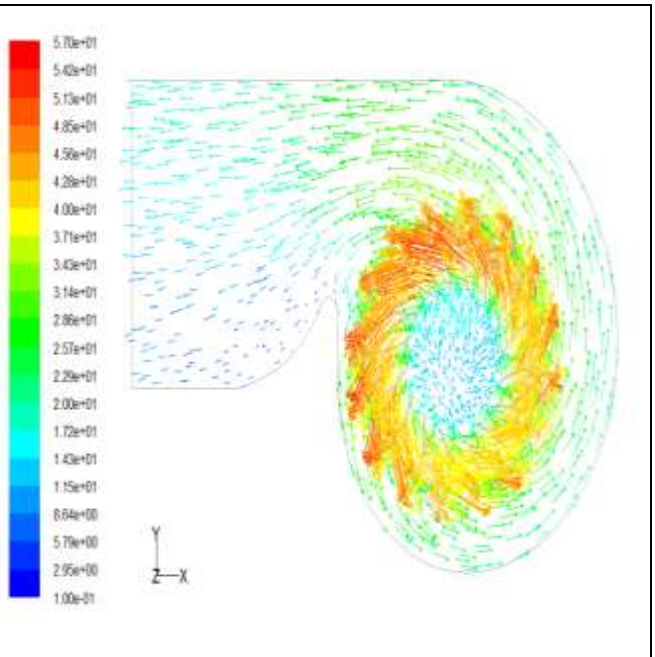

Figure 4.4 (h) shows velocity (in m/s) vectors at mid-plane

Fig. 4.4 (g) shows static pressure blade loading over suction and pressure sides of each blade. This is seen on entire blade area of all 16 number of blades mounted on impeller. Even pressure distribution is seen on suction and pressure side of each blade.

Fig. 4.4 (h) shows velocity vectors at mid-plane. Streamlined flow pattern is observed. Little recirculation of flow is seen within impeller blade passage. Recirculation and stagnation of flow is also observed near tongue region [18, 32].

Similar way results are obtained for forward curved radial tipped (FCRT) centrifugal fan and shows similar trend seen for backward curved radial tipped (BCRT) fan and traces identical pattern of standard fan performance.

Chapter 5
EXPERIMENTAL INVESTIGATIONS

Experiments are carried out to find optimum number of blades and slip factor. Later, all the fans fabricated as per individual and unified design methodologies, are tested experimentally to get their performance characteristics. These tests are conducted according to standard test code IS: 4894-1987, Indian Standard Specification for Centrifugal Fans (First Revision), Reaffirmed 1994 [9]. Precise and calibrated measuring and sensing instruments are used to check their output in terms of head, capacity, and efficiency.

Complete experimental and measurement work is divided in to five phases. All phases are individually described below:

5.1 Phase - I: Experimental Optimization of Finite Number of Blades under Varying Speed Conditions

Suction pressures are varied with the help of different sized orifice plates at inlet airway duct. Blades are screwed between front and back shroud plates in such a way that removal and fixation of blades can be done with ease and without disturbing its dynamic balancing. Blades can be varied in step of 8, 12, 16 and 24. Fan performance is evaluated under varying number of blades at different conditions of suction pressure. Entire fan assembly is made from transparent thermoplastic acrylic sheets for better flow visualization. This phase of experiment is sub-divided into two stages. Both stages are under varying speed conditions.

5.1.1 Stage 1: Influence of Suction Pressure on Performance of the Fan

During this stage 1 of phase 1 speed of rotation is kept constant at design point. Suction pressures are varied with the help of different sized orifice plates. Varied suction pressure lies in the range of 98 to 1374 N/m^2.

5.1.2 Stage 2: Optimization of Finite Nos. of Blades

This stage of experiment is targeted towards optimization of finite nos. of blades. 24 impeller blades are screwed between front and back shroud plates. Number of blades is varied in 4 steps as 8, 12, 16 and 24. Blades are varied in such a way that dynamic balance does not get disturbed in each case. Readings are recorded for each set of number of blades under varied speed conditions at optimized suction pressure as achieved in above stage.

Fig. 5.1 shows transparent fan and test set up with measuring instruments used in stage 1.

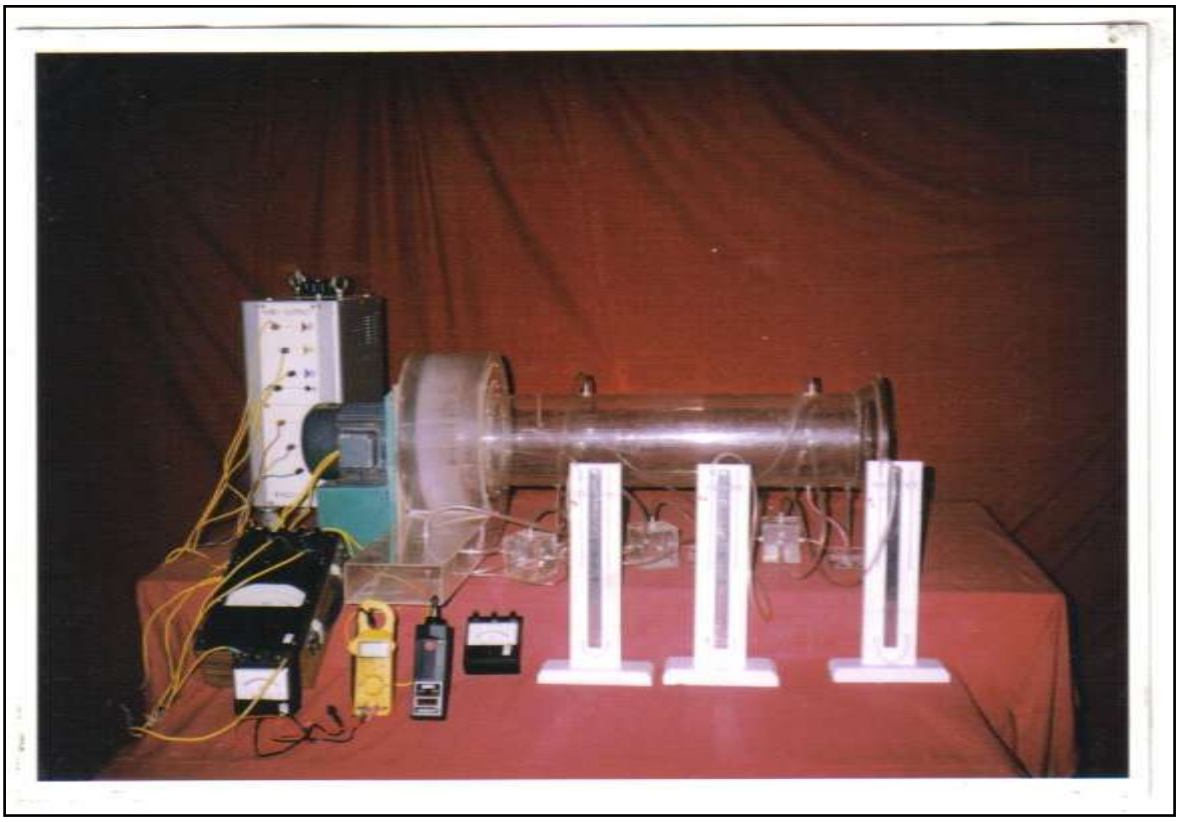

Fig. 5.1 Transparent Fan with Experimental Set Up

5.2 Phase - II: Experimental Investigation on Slip Factor at Varying Number of Blades Condition

Slip loss is defined as the ratio of actual & ideal values of the whirl components at exit of impeller. It has significant effect on fan performance. Experiments are made on above referred transparent test setup of phase 1 with minor modifications.

A specially designed and calibrated sturdy three-hole probe having 0.7 mm internal diameter is used for measuring and sensing local velocities. Probe readings are taken at all A to H insertion locations as per location plan given in fig 5.2. Probe is traversed axially along blade width to take seven different sub location readings per insertion location.

Flow angles at exit are measured with relative and absolute velocity vectors. These parameters are used to construct actual and theoretical velocity diagrams and henceforth to calculate local slip factors. Slip factor is measured in all volute locations (A to H) for 12, 16 and 24 number of blades over entire blade width.

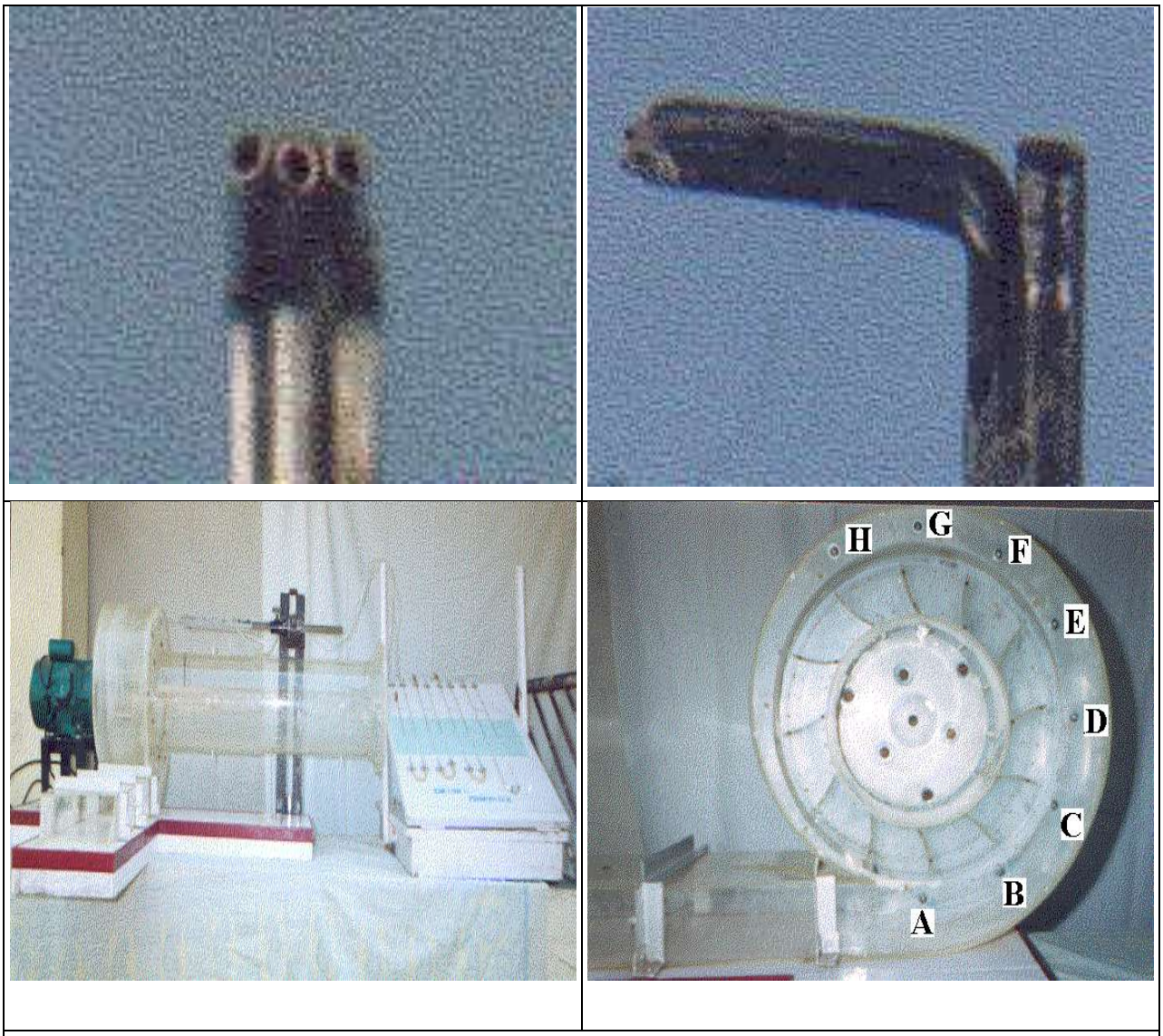

Fig. 5. 2 Photographic views of 3 hole wedged probe with Probe Insertion Locations

5.3 Phase - III: Comparative Assessment of Design Methodologies

Forward curved radial tipped impeller centrifugal fans are fabricated as per design methodology traced out by using fundamental principles of fluid flow [1, 3, 4, 5, 6] and other design methodologies suggested by Church A. H. [7] and Osborne W. C. [8] Fans are fabricated using 14 gauge (2.03 mm) M.S. sheets. Proper care is taken to avoid distortion during welding. Impellers are dynamically balanced and outlet damper is used for mass flow variation. The speed of impeller rotation is varied in steps of 500, 1000,1500,2000,2500, and 2800 rpm.

Extensive experimental investigations are made to evaluate individual design methodology suggested by different researchers. Measurements are made for fan inlet and outlet pressures, volute casing pressure distribution (at 90^0,120^0,180^0,240^0 and 300^0), input power and average air velocity at different speed and damping conditions. This facilitates to

find actual stage pressure head developed across the fan, average air discharge, shaft power and airpower developed during each set of observation.

Based on experimental results obtained, it is recognized that there exists a wide performance difference amongst fans under study. All fans are not performing as per mark. It has revealed that there is a need to develop unified design methodology.

5.4 Phase - IV: Unified Design Methodology and Comparative Performance Evaluation of Forward and Backward Curved Radial Tipped Centrifugal Fan

Experimental evaluation of fans stated in phase III has revealed that there is wide performance variation exists. Hence Successful outcomes of fundamental [1, 3, 4, 5, 6], Church [7] and Osborne [8] designs are incorporated and unified design methodology for radial tipped centrifugal fan is developed. This is design optimization process for a radial tipped centrifugal fan/blower.

This phase of research work is planned for performance evaluation of forward and backward curved impeller fans fabricated as per unified design. All the fabrication work is carried out by using mild steel sheets. The special fixtures are made for achieving desired blade profile & casing to avoid distortion after welding. Impellers are rotating parts, so they are dynamically balanced after fabrication. Tests are conducted according to test code IS: 4894-1987 [9].

Fig 5.3 shows forward and backward Curved Radial Tipped Impellers made as per unified design methodology and the test setup. Measurements are recorded as explained in previous section.

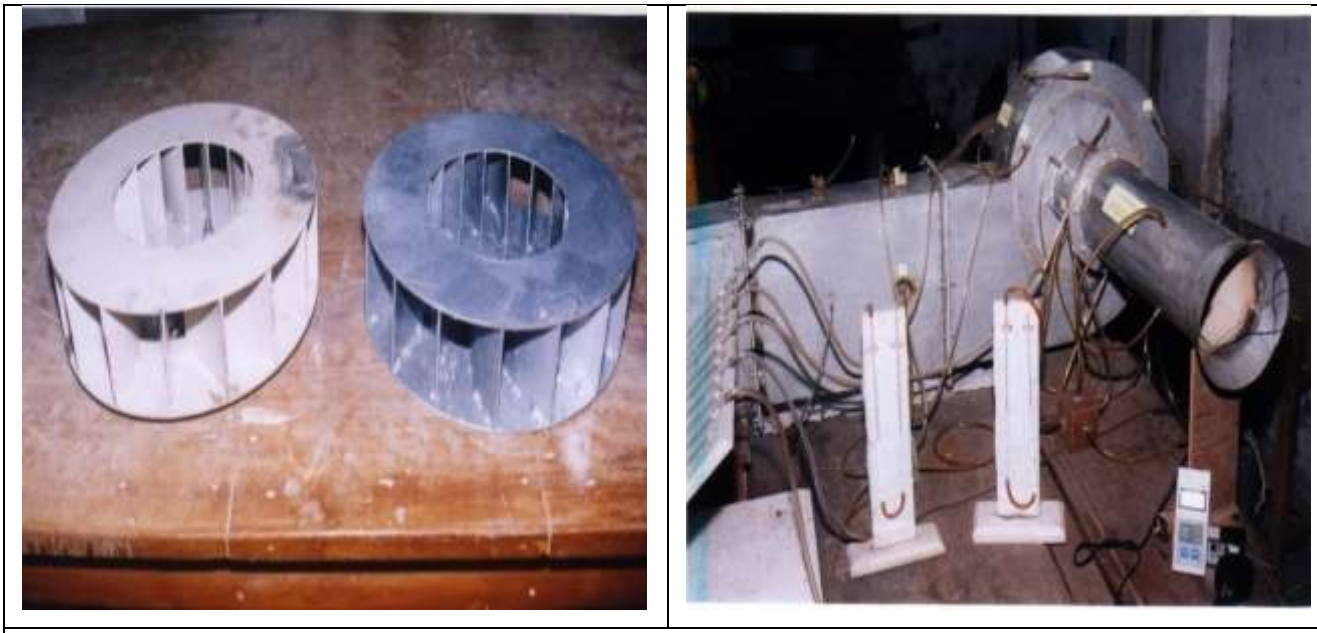

Fig. 5. 3 Forward And Backward Curved Radial Tipped Impellers And Test Setup

5.5Phase - V: Assessment of Theoretical and Experimental Losses

Losses evaluated by different methods and proposed by various researchers differ widely. Experiments are carried out to measure hydraulic, leakage, mechanical losses and corresponding efficiencies. Comparative analysis is of theoretical and experimental values of various losses occurring in radial tipped centrifugal fan are made.

Centrifugal fans with 8 inch sized forward and backward curved radial tipped impellers as per unified design are fabricated from transparent material. Static and velocity pressure heads are measured by specially developed and calibrated five hole probe, at design and off design speed conditions. Other parameters are recorded simultaneously. Entry and exit impeller velocity triangles are prepared at different peripheral locations. Slip factor is calculated at all volute locations for different speed conditions.

Fig. 5.4 and 5.5 shows five hole pressure probe and experimental setup showing various probe insertion locations.

Fig. 5.4 Five Hole Pressure Probe

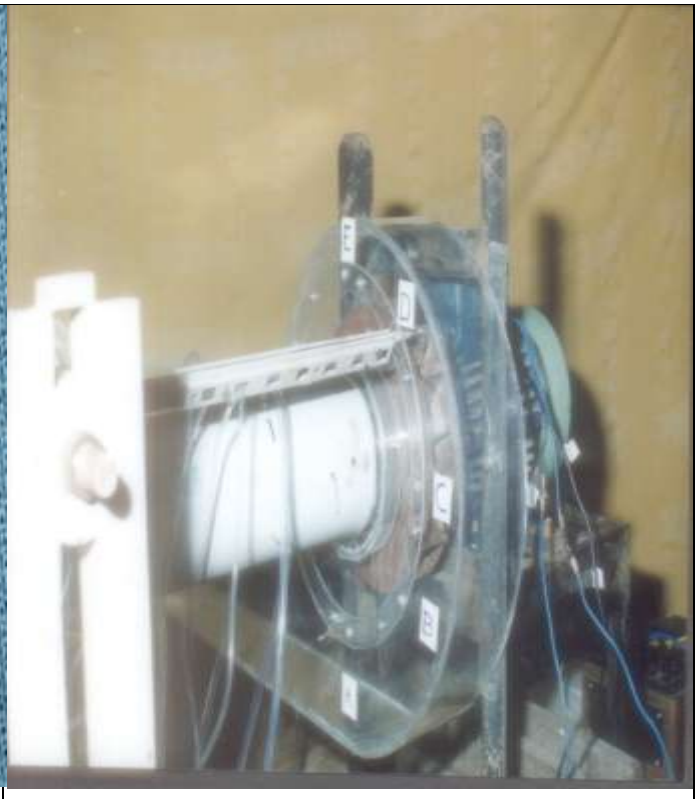

Fig. 5. 5 Test Set Up Showing Various Positions of Probe Insertion

Uncertainty Analysis

An uncertainty analysis is carried out according to Kline and Meclintock method compiled by Hollman J.P. [15]. All uncertainties in measurements are observed to be well within ± 5.0 % of design point values.

Chapter 6

RESULTS & DISCUSSIONS

Extensive experimental investigations are made to find optimized design methodology for radial tipped centrifugal fan. The experiments are made in five phases as explained in chapter 5. Performance of each phase is critically evaluated. Salient features of results obtained during this course work are discussed herewith in five phases.

6.1 Phase - I: Experimental Optimization of Finite Number of Blades under Varying Speed Conditions

This phase of experiment is sub-divided into two stages. Both stages are under varying speed conditions. Fan performance is evaluated for both stages.

6.1.1 Stage 1: Influence of Suction Pressure on Performance of the Fan

Figure 6.1 and 6.2 presents the distinctive results of influence of suction pressure on discharge for 16 and 24 numbers of blades. It is observed that when suction pressure increases, the discharge gradually increases and achieves maxima of 0.3304 m^3/s at 530 N/m^2 suction pressure, then after it gradually reduces with further increase in suction pressure.

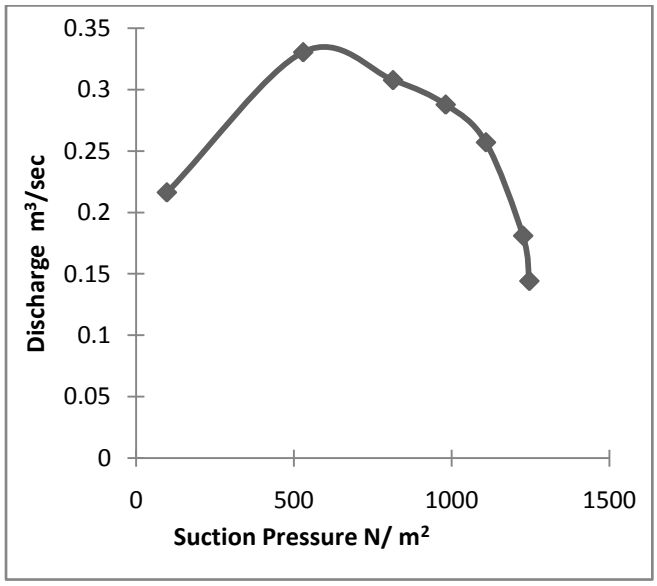

Fig. 6.1 Suction Pressure v/s Discharge at 24 number of Blades

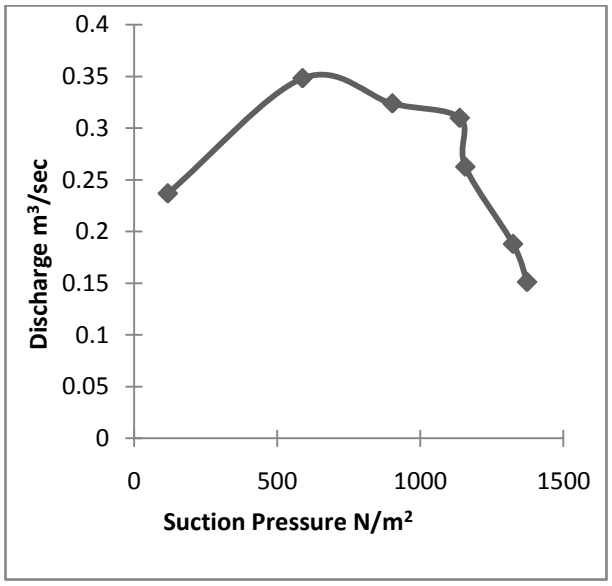

Fig. 6.2 Suction Pressure v/s Discharge at 16 number of Blades

This behavior is standard fan characteristic. When suction resistance increases, losses increases and discharge begins to drop after achieving maxima. [1, 7, 8]

Best efficient performance of this fan is achieved at a static stage pressure rise of 882.9 N/m^2 at discharge of 0.288 m^3/s and total stagnation efficiency of 87.6%. This performance is quite away from the design point discharge. This underlines the need for critical evaluation of design equations for suction side. Above exercise is repeated for 8 and 12 number of blades.

6.1.2 Stage 2: Optimization of Finite Nos. of Blades

Table 6.1 summarizes optimum performance parameters obtained during stage 1 for each set of number of blades and varying suction pressure conditions at design speed.

Table 6.1 Optimum Performance Parameters as a Function of Number of Blades

No. Of Blades	Pressure Developed N/m^2		Stage Efficiency Optimum		Max. Discharge m^3/s
	Δp_{Static}	$\Delta p_{Stagnation}$	η_{Static}	$\eta_{Stagnation}$	
8	1383.2	1580.5	55.1	75.7	0.340
12	1324.4	1443.0	55.7	79.8	0.327
16	**1432.3**	**1637.3**	**60.5**	**88.2**	**0.348**
24	1275.3	1393.9	69.1	87.6	0.330

Study of these results clearly indicates that the best performance is achieved under 16 numbers of blades. This has attributed to the fact that with 16 nos. of blades, the flow is efficiently guided without separation having lower frictional losses [2, 21].

During experiments of stage 1, suction pressure is optimized for 110 mm orifice plate. At stage 2, performance parameters obtained for each set of number of blades at varied speed in the steps of 500, 1000, 1500, 2000, 2500 and 2950 rpm design speed at constant optimized suction pressure. Results are presented in Table 6.2.

Table 6.2 Optimum Performance Parameters at Varied Speed Conditions

No. Of Blades	Pressure Developed N/m^2		Stage Efficiency Optimum		Max. Discharge m^3/s
	Δp_{Static}	$\Delta p_{Stagnation}$	η_{Static}	$\eta_{Stagnation}$	
8	1098.7	1489.7	55.1	74.7	0.262
12	981.0	1345.9	55.7	75.3	0.256
16	**1128.2**	**1522.4**	**56.8**	**76.7**	**0.263**
24	1010.4	1388.0	60.7	83.4	0.257

From table 6.1 and 6.2, it is observed that most of optimum performance parameters are achieved with 16 nos. of blades. However design point discharge is not achieved even with this optimum 16 number of blades. Optimization of number of blades of centrifugal fan impeller involves a maximization problem of multivariable functions [21], and hence further research work needed to trace out better design methodology.

The head coefficient, power coefficient and discharge coefficient obtained for this case at 16 numbers of blades are 0.00841, 2.470 and 0.289 respectively.

6.2 Phase - II: Experimental Investigation on Slip Factor at Varying Number of Blades Condition

Slip factor is defined as the ratio of actual exit whirl velocity to the theoretical exit whirl velocity. Theoretical (ideal) value of whirl velocity at exit of blade is obtained analytically while the actual value of whirl component is obtained experimentally. Experimental values of whirl components are measured at 'A to H' volute locations as given in Fig. 5.2 and explained in section 5.2. Each volute location is further divided in seven sub- locations axially. Slip factor at each location is calculated by averaging sub location slip factor values.

Comparative assessments of experimental and empirically correlated slip factor values given by BALJE, STODOLA & STANITZ [1, 3] are given in table 6.3 for 16 numbers of blades.

Table 6.3 Comparative Evaluations of Slip Factors for 16 Numbers of Blades

Deg.	Location	Av. Slip factor (Experimental)	Average Of All Locations (Experimental)	Theoretical Values Using Co-Relations		
				Balje	Stodola	Stanitz
0	A	0.821	0.771	0.780	0.804	0.876
30	B	0.756				
60	C	0.743				
90	D	0.734				
120	E	0.735				
150	F	0.792				
180	G	0.814				
210	H	0.773				

Fig. 6.3 shows graphical presentation of empirical and experimental slip factor at various volute casing locations for 16 numbers of blades. While Fig. 6.4 shows

experimental and empirical values of slip factor for radial tipped centrifugal fan under study with respect to number of blades.

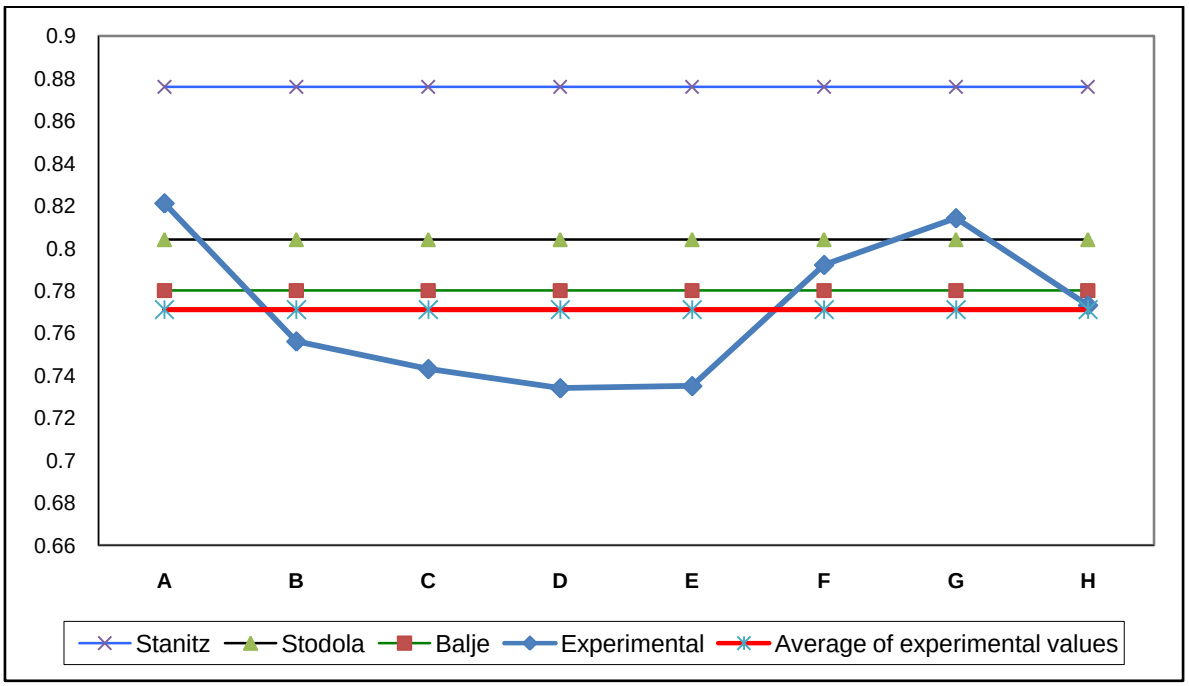

Fig. 6.3 Slip factor v/s. Volute Casing Location for 16 number of Blades

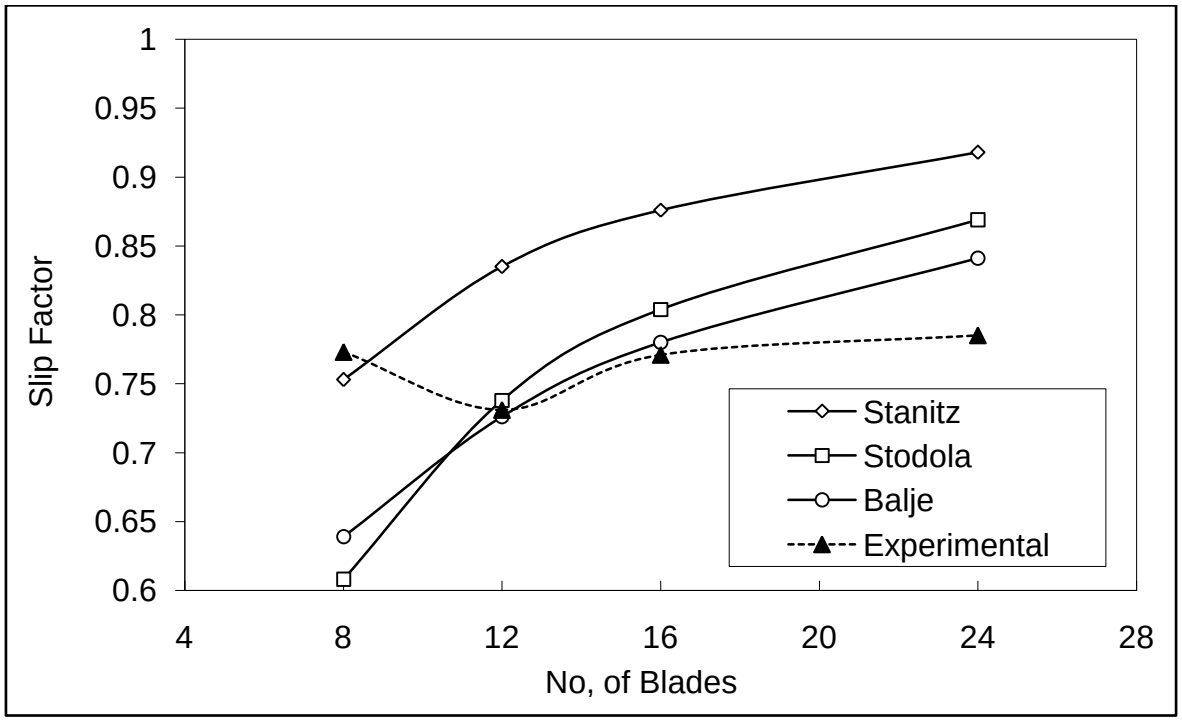

Fig. 6.4 Graphical Comparison of Slip Factors for 8, 12, 16 and 24 Number of Blades

Absolute velocity components and whirl angles are measured experimentally with the help of angular movement mechanism provided on sliding fixture. Fig. 6.5 shows impeller exit velocity diagram at ‘A to H’ locations for 16 number of blades.

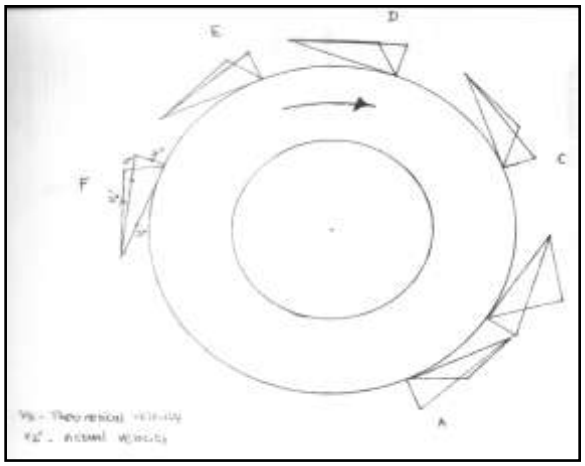

Fig. 6.5 Typical Velocity Triangle at Different Locations for 16 Numbers of Blades

Based on Fig. 6.3 and 6.4, it is observed that experimental value of slip factor is found 3 to 12% less with respect to various empirical co-relations. This difference is maintained for almost all set of number of blades. Exceptions are seen at very low number of blades due to sudden increase of turbulence between two blades. This supports work of Yedidiah Sh. [10] of presenting a new model of slip factor that resolves basic discrepancies observed between old theories.

It is also observed that slip factor is varying with number of blades. Hence it can be said that slip factor is not only dependent of flow. Similar is supported by research work of R. Ajithkumar [11] as concluded that slip factor is a function of number of vanes, diameter ratio, and outlet blade angle and flow conditions after impeller.

Lal & Vasandani [23] has studied slip factor effect on designing of impeller. They concluded that slip factor reduces due to non-uniform velocity distribution at impeller exit. This is also confirmed from table 6.3 and Fig. 6.5.

It is also seen from experimental results that slip factor profile over blade width at impeller exit shows negative parabolic profile. Hence design of exit blade section must be parabolic to improve blade tip slip factor.

6.3 Phase - III: Comparative Assessment of Design Methodologies

Extensive experimental investigations are made to evaluate three design methodologies traced out by using fundamental principles of fluid flow [1, 3, 4, 5, 6] and others suggested by Church A. H. [7] and Osborne W. C. [8] Performance of each design methodology is critically evaluated and discussed in subsequent sections.

6.3.1 Fundamental Design [1, 3, 4, 5, 6]

Fig. 6.6 and 6.7 shows the fundamental design performance obtained in terms of discharge v/s stage pressure rise and stagnation total efficiency.

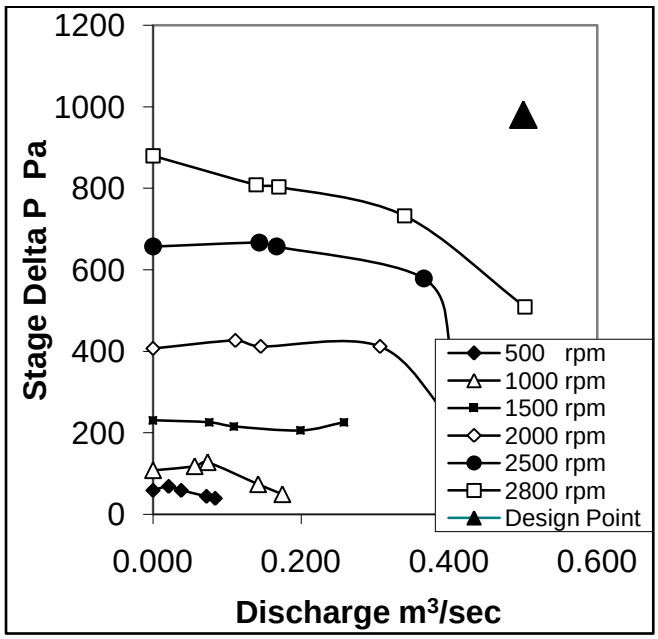

Fig. 6.6 Discharge V/s Stage Pressure Rise

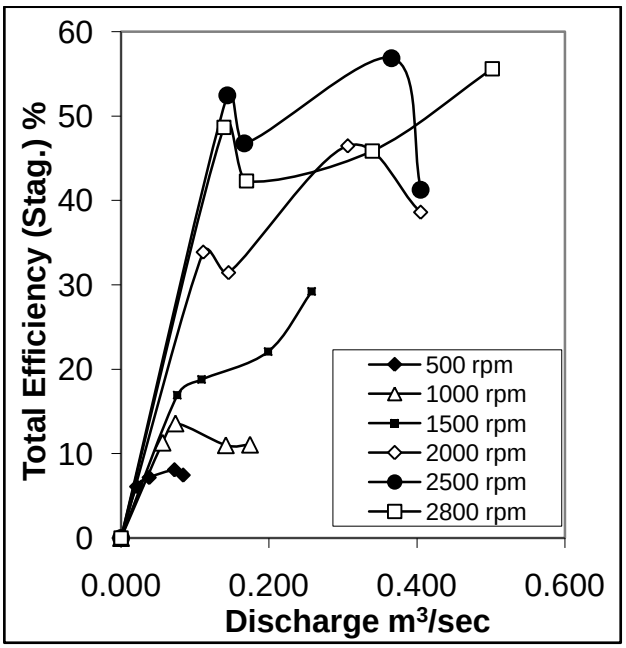

Fig. 6.7 Discharge V/s Stagnation total Efficiency

It is observed that fan inlet pressure, measured very near to impeller inlet is – 488 Pa while fan outlet pressure is only 20 Pa. Thus, total static pressure rise of 509 Pa is achieved at 2800 rpm and 0% damping. The discharge obtained is 0.502 m^3/s and the total efficiency at this point is 56%. It is interesting to note that only discharge is achieved as per design point.

It can be said for fundamental design that, it is better for suction pressure development but very much lacking in generating outlet pressure. This design can achieve only design flow rate at maximum speed and 0% damping. Best operating range is not possible to trace. Present efficient operating range for this fan is at 0 to 25% damping position for all speed conditions. At off-design operating points, discharge gets very much reduced.

It is suggested that proper loss estimation and redesigning for impeller outlet diameter can help to achieve design point performance. In addition to this, increase in nose radius may also be helpful in achieving higher outlet pressure with decrease in discharge loss due to re-circulation.

6.3.2 Church Design [7]

Results obtained under Church design are plotted in fig 6.8 and 6.9.

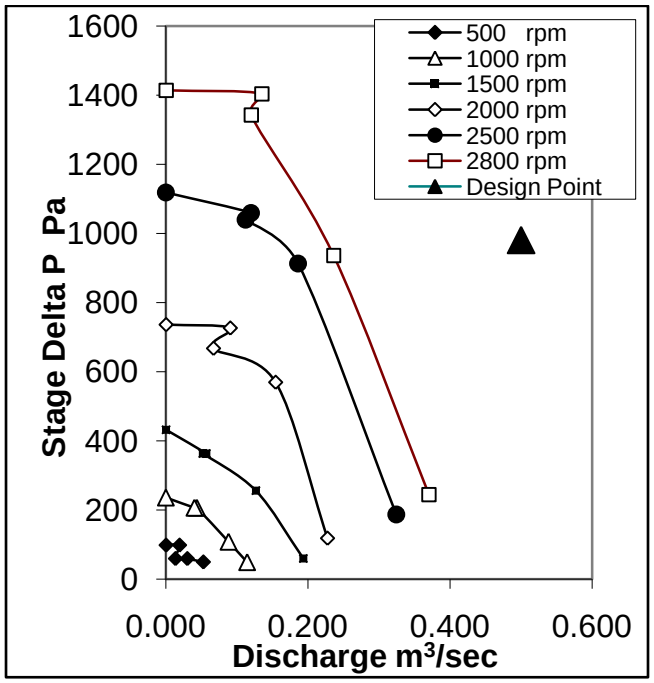

Fig. 6.8 Discharge V/s Stage Pressure Rise

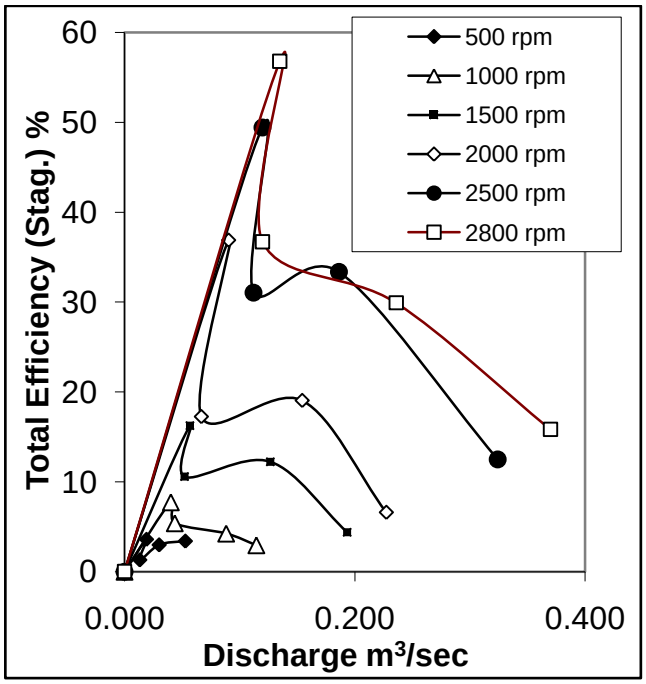

Fig. 6.9 Discharge V/s Total Stagnation Efficiency

As per this design methodology, the experimental results are quite passive. It is seen that fan inlet static pressure, measured very near to impeller inlet is – 10 Pa only, while fan outlet pressure is 1393 Pa. Thus, total static pressure rise of 1404 Pa is achieved at 2800 rpm and 75% damping. The discharge obtained is 0.135 m^3/s and the total efficiency at this point is 57%.

Performance has shown a need for redesigning of suction dimensions. Impeller inlet dimensions along with blade width should increase so that higher flow rate can be achieved.

Church has used pressure coefficient K'. Its value lies empirically between 0.5-0.65. This is used to calculate impeller outlet diameter. This empirical concept requires reconsideration.

However if design point performance is neglected, the best operating region for this design lies between 50 to 75% damping conditions.

6.3.3 Osborne Design [8]

The results obtained under this design are very much encouraging. Major performance parameters achieved are on higher side of design point. This is useful in establishing best operating range of any turbo machine.

Fig. 6.10 and 6.11 shows the performance obtained under Osborne design.

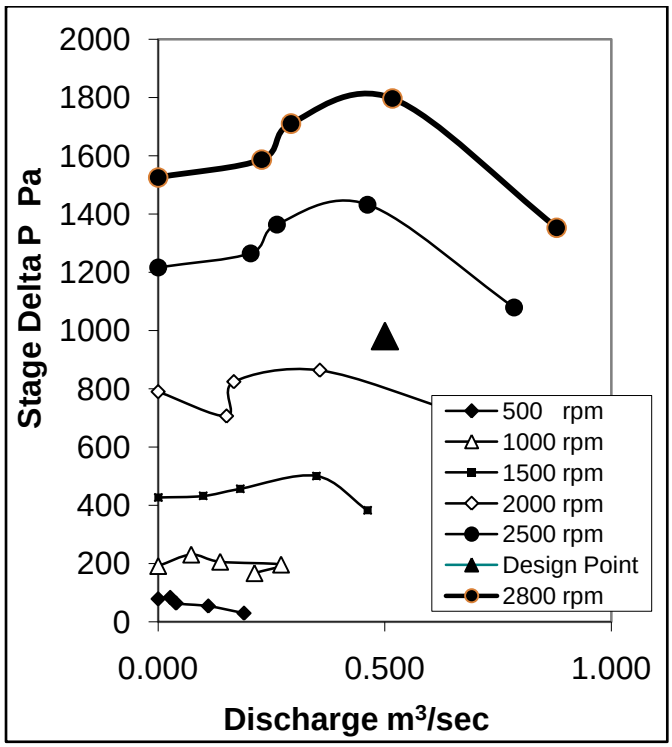

Fig. 6.10 Discharge V/s Stage Pressure Rise

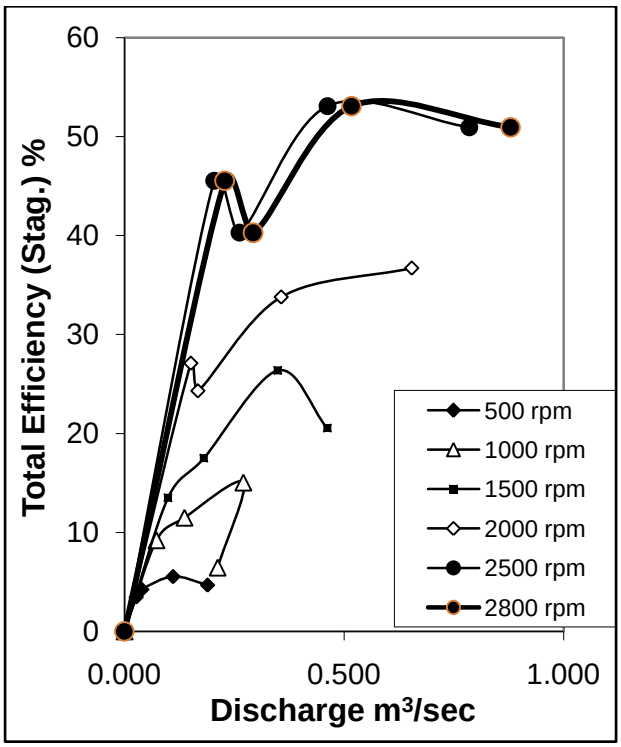

Fig. 6.11 Discharge V/s Total Stagnation Efficiency

As per this design methodology, the experimental results are positive. It is observed that fan inlet static pressure, measured very near to impeller inlet is – 345 Pa while fan outlet pressure is 1452 Pa. Thus, total static pressure rise of 1796 Pa is achieved at 2800 rpm and 25% damping. The discharge obtained is 0.517 m^3/s and the total efficiency at this point is 53%.

Based on results obtained, it can be said that all major performance parameters like inlet pressure, outlet pressure, volume flow rate, stage pressure rise etc. are achieved as per design point. This design is more suitable where higher suction head is required. Flow regulation is possible to run fan at off design conditions, too.

Major limitation of this fan is consumption of higher input power compared to others two. It is also big in size compared to fundamental and Church design. To overcome these limitations, conservative loss estimation may help while redesigning. The best operating range for this fan is at 0 to 25% damping and 2800 rpm.

6.3.4 Comparative Performance Assessment of Design Methodologies

Table 6.5 shows comparative assessment of efficient operating point performance obtained under design methodologies described above for design point discharge 0.5 m^3/s and 981 Pa stage pressure rise. This is 466 watts developed airpower at 2800 rpm.

It is worth noting here that as per table 3.1, the design dimensions as per Fundamental [1, 3, 4, 5, 6], Church [7] and Osborne [8] designs have wide variations. Similarly their performance is also varying. Volumetric flow of fundamental and Church design is very much less as a result of recirculation. Same is confirmed by R. C. Worster [18].

Table 6.5 Comparative Performance Assessment of Design Methodologies

Design	Speed rpm	Damp.	Fan Delta P Pa	Avg. Air Discharge m^3/s	Static Air Power Watts	Input Power Watts	Total Eff. %
Design Point	2800	0%	981	0.500	466		--
Fundamental	2800	0%	509	0.502	256	680	56
Church	2800	75%	1404	0.135	190	338	57
Osborne	2800	25%	1796	0.517	929	1883	53

It is found that the fluid deviates and becomes more non-uniform at impeller outlet when there is reduction in volume flow rate. Same is confirmed earlier by Prasad, Ganeshan & Prithviraj [19]. This comparative assessment also revealed that fundamental design requires improvements for increase in stage pressure head while Church design requires improvements for discharge to meet design point performance.

Osborne design requires redesigning for controlling stage pressure rise and reduction in input power. These improvements in design will lead to increase of total efficiency at design point flow and pressure rise.

This performance assessment has indicated that there is a need to develop unified design methodology for radial tipped centrifugal fan/blower to get real design point performance.

6.4 Phase - IV: Unified Design Methodology and Comparative Performance Evaluation of Forward and Backward Curved Radial Tipped Centrifugal Fan

Unified design procedure is developed from fundamental concepts and involving minimum assumptions. Forward and backward curved radial tipped impeller fans are fabricated as per this unified design methodology. Their performance is measured as per standard test procedure IS: 4894-1987, Indian Standard Specification for Centrifugal Fans.

Fig. 6.12 to 6.15 shows the performance curves obtained for forward and backward curved radial tipped centrifugal fans.

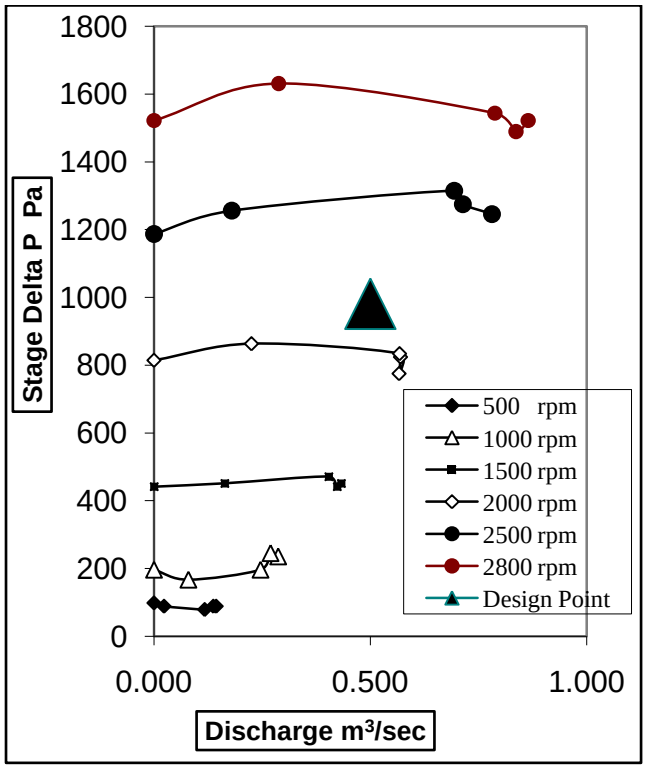

Fig. 6.12 Discharge V/s Stage Pressure Rise (FCRT)

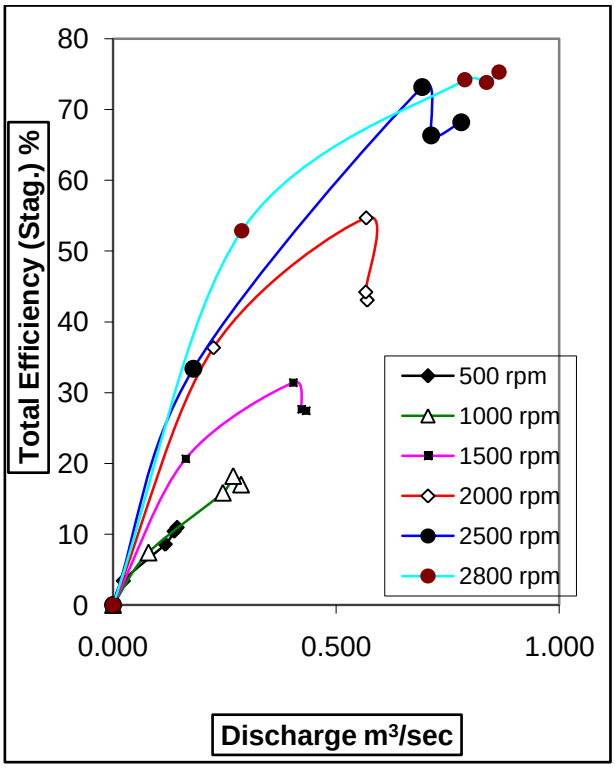

Fig. 6.13 Discharge V/s Static Efficiency (FCRT)

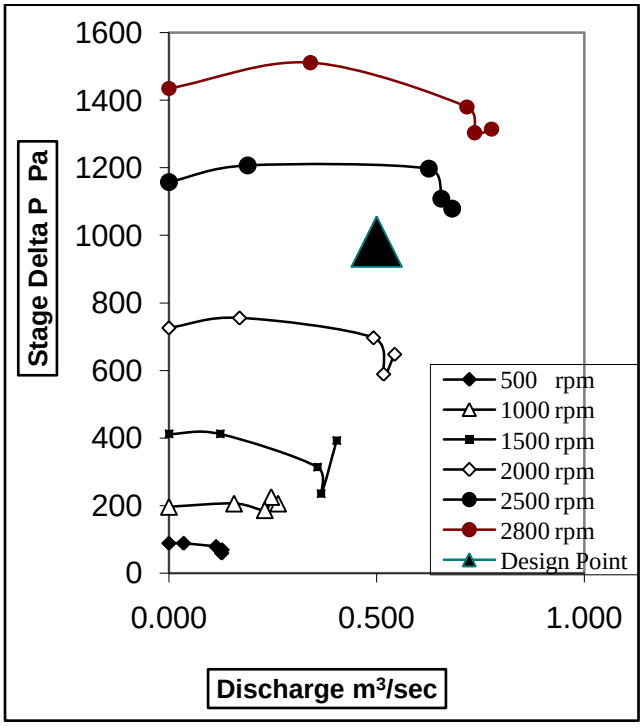

Fig. 6.14 Discharge V/s Stage Pressure Rise (BCRT)

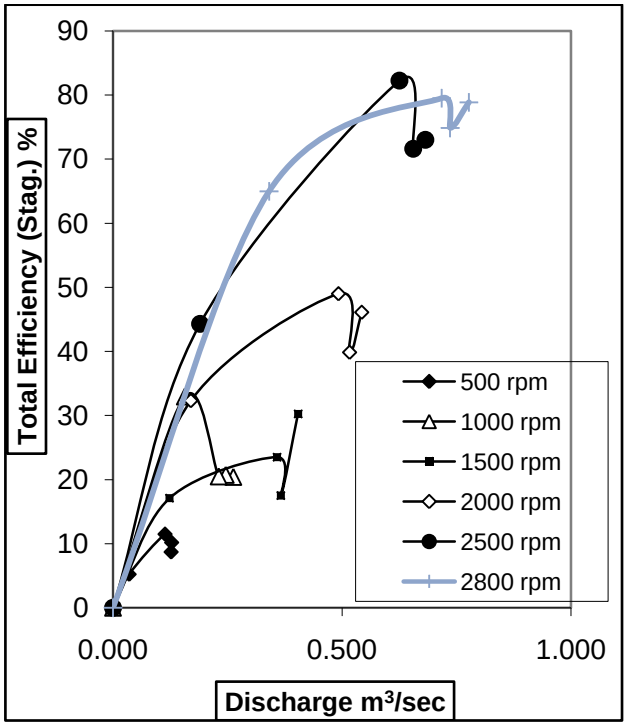

Fig. 6.15 Discharge V/s Static Efficiency (BCRT)

Table 6.6 summarizes the results of optimum operating range received.

Table 6.6 Optimum Operating Ranges for Forward and Backward Curved Radial Tipped Fan as per Unified Design

Blade	RPM	Damping	Static Stage Pressure Rise Pa	Discharge m^3/s	Air Power Watts	Input Power Watts	Total Efficiency %
	2800	Design Point	981	0.5	466		
FCRT	2800	50%	1544	0.787	1215	1746	74
FCRT	2800	75%	1632	0.288	471	896	53
BCRT	2800	50%	1380	0.717	990	1321	80
BCRT	2800	75%	1512	0.340	514	802	65

These results clearly show that fan based on unified design is good enough to achieve desired performance. Major performance parameters achieved are on higher side of design point. Flow regulation is possible between the operating ranges of 50 to 75% damping at 2800 rpm. This can provide flexibility to run fan at off design conditions, too. This fan is consuming less input power compared to Osborne design and at the same time gives better efficiency.

Similarly the results obtained for backward curved radial tipped centrifugal fan are also very much encouraging. The best operating range is obtained between 50 to 75% damping in this case, too. Comparative evaluation of FCRT and BCRT fans has confirmed that total efficiency of BCRT is higher with respect to FCRT fan as given in literature [6].

Thus, it may be stated that unified design methodology outlined during the course of present work may be accepted as the experimentally validated design for radial tipped centrifugal fan, which can confidently offer design point performance.

6.5 Phase - IV: Assessment of Theoretical and Experimental Losses

During this course of work, Centrifugal fans with 8 inch sized forward and backward curved radial tipped impellers as per unified design are fabricated from transparent material. Static and velocity pressure heads at design and off design speed conditions are measured by specially developed and calibrated five hole probe. Other parameters are recorded simultaneously. Entry and exit impeller velocity triangles are prepared at different peripheral locations. Slip factor is calculated at all volute locations for different speed conditions. The average value of slip factor is 0.67. This is less than the empirical value of 0.8 considered for design purpose. This indicates that there is more deviation in actual and the theoretical flow direction. Experimental value of slip factor is found 16% less as

compare to empirical values. Thus this reaffirms the results of experiments carried out on 12 inch size impeller fan by 3 hole probe as described in phase II.

More deviation of exit velocities is observed in small size impellers. It confirms that shorter blade passage height (r_2-r_1) will produce more slip. This is due to absence of proper flow guidance within blade passage. Blade height is important parameter and should be considered while designing. Slip factor have closer agreement at design point conditions but deviates at off design conditions. This confirms study of Mohamad Memardezfouli, Ahmad Nourbakhsh [29].

Further experiments are carried out to measure hydraulic, leakage, mechanical losses. Among all hydraulic losses, impeller losses are found 68%. This confirms the study of Andre Kovats [12] and R J Kind [13]. Volute losses are observed 31 %. Duct friction losses are negligible being less than 1%.

Leakage losses are 15% of actual discharge, while mechanical losses are observed 12% of input power. Comparison of theoretical and experimental losses is presented in Fig. 6.16. Total experimental losses are presented by pie chart in Fig. 6.17.

These facts lead to the conclusion that impeller and volute losses are to be minimized to improve hydraulic efficiency of fans. Impeller losses can be reduced by adopting smaller diameter ratio (d_1/d_2) and tapered shroud plates to reduce rate of diffusion and increase slip factor. Vaned diffuser will help to reduce eddies in volute casing and hence reducing volute losses [1].

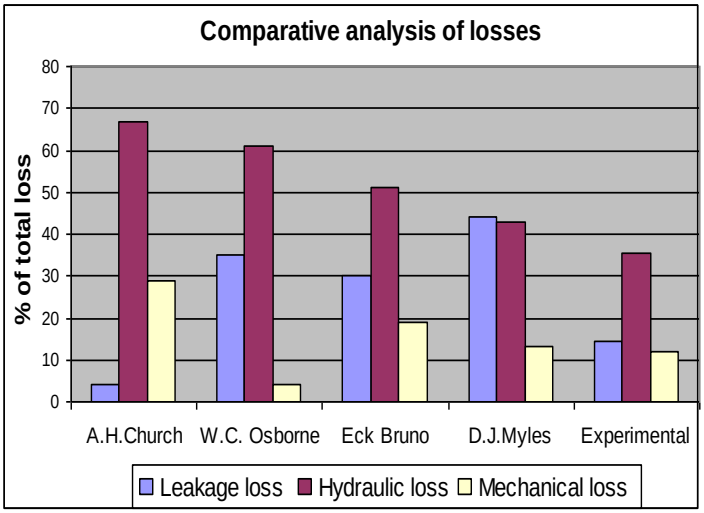

Fig. 6.16 Comparative Analysis of Theoretical and Experimental Losses

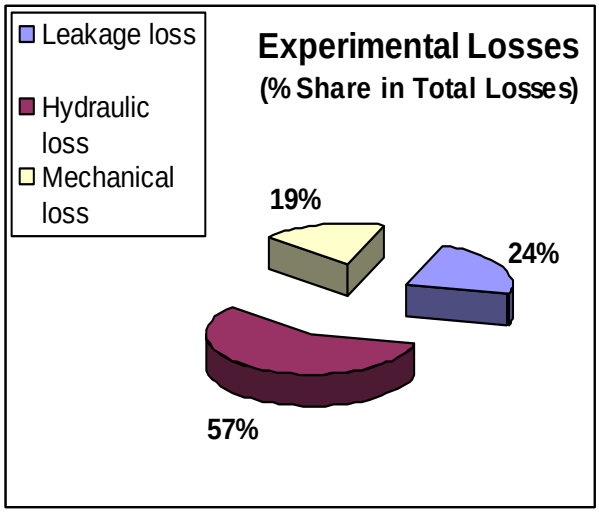

Fig. 6.17 Distribution of Experimental Losses

Similar way, results are obtained for forward curved radial tipped (FCRT) centrifugal fan and shows similar trend seen for backward curved radial tipped (BCRT) fan.

Chapter 7

CONCLUSIONS

Extensive theoretical and experimental investigations carried out during the course of this study, clearly leads to the following conclusions.

(1) The optimum performance of the fan in phase I to optimize finite number of blades is achieved for 16 nos. of blades. This is confirmed under varying suction pressure and varying number of blades conditions at design and off design speed operations. The optimum values of head coefficient, power coefficient and discharge coefficient obtained are 0.00841, 2.470 and 0.289 respectively for 16 nos. of blades. These non dimensional parameters may be used as design guidelines for a radial tipped forward swept centrifugal fan.

(2) Experimental investigations made by 3 hole and five hole probes on slip factor with varying number of blades has lead to following conclusions;

- A comparative assessment of experimental and empirically correlated slip factor values is made. Theoretical values using co-relations given by BALJE, STODOLA & STANITZ [1, 3] are 0.780, 0.804 and 0.876 respectively, while average value of experimental slip factor is 0.771.
- It is observed that experimental value of slip factor is found 3 to 16% less with respect to various empirical co-relations. This difference is maintained for almost all set of number of blades. Exceptions are seen at very low number of blades due to increase of turbulence between two blades. It confirms literature evidence of Lal & Vasandani [23].
- Empirical correlations suggest constant value of slip factor over entire exit blade width. But it is contradicted by experimental results that slip factor along blade width in vane to vane and meridional plane of impeller is not constant. Experimental value of slip factors for 16 numbers of blades at 2800 rpm along blade width varies in the range of 0.633 to 0.788. It varies due to boundary layer thickness and due to changes in exit components of whirl velocities at different blade locations.
- Empirical correlations available in literature states that slip is a function of impeller alone and is not affected by flow rate. Empirical correlations are also

neglecting effects of blade width, volute casing, impeller geometry and specific speed. This is misleading concept. Existence and effect of above referred parameters is confirmed by this experimental study. Evidences of present experimental results supports work of Yedidiah Sh. [10] of presenting a new model of slip factor which resolves basic discrepancies observed between old theories.

- R. Ajithkumar [11] has concluded in his research work that slip factor is a function of number of vanes, diameter ratio, and outlet blade angle and flow conditions after impeller. It is reaffirmed by figure 6.4 showing variation of slip factor with respect to number of impeller blades.
- Lal & Vasandani [23] has studied slip factor effect on designing of impeller. They concluded that slip factor reduces due to non-uniform velocity distribution at impeller exit. This is confirmed by table 6.3 and Fig. 6.5.
- It is also seen from experimental results that slip factor profile over blade width at impeller exit shows negative parabolic profile. Hence design of exit blade section must be of parabolic shape to improve blade tip slip factor.
- Slip factor have good agreement at design point conditions but deviates at off design conditions. This confirms study of Mohamad Memardezfouli, Ahmad Nourbakhsh [29].

(3) Literature review has revealed that there exists a lacuna towards experimentally validated fan design. To fill this void, experimental investigations are made and performance obtained under specific design methodologies concluded as following;

- It is observed for fundamental design [1, 3, 4, 5, 6] that fan inlet static pressure, measured very near to impeller inlet is – 488 Pa while fan outlet static pressure is only 20 Pa. Thus, total static pressure rise of 509 Pa is achieved at 2800 rpm and 0% damping. The discharge obtained is 0.502 m^3/s and the total efficiency at this point is 56%. It is interesting to note that only discharge is achieved as per design point. Fundamental design is better for suction pressure development but very much lacking in generating outlet pressure. This design can achieve only design flow rate at maximum speed and at 0% damping. Efficient operating range for this fan is at 0 to 25% damping position for all speed conditions. At off-design operating points, discharge gets very much reduced. It is suggested that proper loss estimation

and redesigning for impeller outlet diameter can help to achieve design point performance. In addition to this, increase in nose radius may also be helpful in achieving higher outlet pressure with decrease in discharge loss due to re-circulation.

- As per Church [7] design methodology, the experimental results are quite passive It is seen that fan inlet static pressure, measured very near to impeller inlet is – 10 Pa only, while fan outlet pressure is 1393 Pa. Thus, total static pressure rise of 1404 Pa is achieved at 2800 rpm and 75% damping. The discharge obtained is 0.135 m^3/s and the total efficiency at this point is 57%. In Church design, Performance has shown a need for redesigning of suction dimensions. Impeller inlet dimensions along with blade width should increase so that higher flow rate can be achieved.

  Church has used pressure coefficient K'. Its value lies empirically between 0.5-0.65. This is used to calculate impeller outlet diameter. This empirical concept requires reconsideration.

  However if design point performance is neglected, the best operating region for this design lies between 50 to 75% damping at 2800 rpm.

- As per Osborne [8] design methodology, the experimental results are positive. It is observed that fan inlet pressure, measured very near to impeller inlet is – 345 Pa while fan outlet pressure is 1452 Pa. Thus, total static pressure rise of 1796 Pa is achieved at 2800 rpm and 25% damping. The discharge obtained is 0.517 m^3/s and the total efficiency at this point is 53%.

  Based on results obtained, it can be said that all major performance parameters like inlet pressure, outlet pressure, volume flow rate, stage pressure rise etc. are achieved as per design point. This design is more suitable where higher suction head is required. Flow regulation is possible to run fan at off design conditions, too.

  Major limitation of this fan is consumption of higher input power compared to others two. It is also big in size compared to fundamental and Church design. To overcome these limitations, conservative loss estimation may help while redesigning. The best operating range for this fan is at 0 to 25% damping at 2800 rpm.

Based on experimental results obtained for fundamental, Church and Osborne design methodology, it is recognized that there exists a wide performance difference

and all fans are not performing as per mark. It has revealed that there is a need to develop unified design methodology to get design point performance.

(4) Successful outcomes of fundamental [1, 3, 4, 5, 6], Church [7] and Osborne [8] designs are incorporated together and unified design methodology for radial tipped centrifugal fan is developed. Experimental investigations carried out on forward and backward curved radial tipped impeller fans fabricated as per unified design methodology facilitates to draw the following conclusions;

- Both fans run efficiently and gives design point performance at 2800 rpm in damping range of 50% to 75 % as given in Table 6.6. The pressure head generated by forward curve radial tipped centrifugal fan is 1632 Pa. This is higher than 1512 Pa, achieved for backward curved radial tipped centrifugal fan and this confirms that for the requirement of achieving higher pressure head, one can select the forward curved radial tipped centrifugal fan.
- The total stagnation efficiency of backward curved radial tipped centrifugal fan is 80% higher than 74% of forward curved centrifugal fan achieved at same speed and damping conditions. This means that for achieving desired performance at high efficiency, one has to go for backward curved radial tipped centrifugal fan [6].
- FCRT fan offers 0.787 m^3/s discharge and BCRT fan offers 0.717 m^3/s discharge at similar operation conditions. This is achieved on higher side, as compared to 0.5 m^3/s design point discharge. This fan is consuming less input power compared to Osborne design and at the same time gives better efficiency.

Thus, it may be stated that unified design methodology outlined during the course of present work may be accepted as the experimentally validated design for radial tipped centrifugal fan, which can confidently offer design point performance.

(5) Losses proposed by various researchers differ widely. Hence, experimental investigations on losses are made to ascertain the performance of radial tipped centrifugal fan. The impeller losses are major contributor and found 68%of total hydraulic losses. This confirms the study of Andre Kovats [12] and R J Kind [13]. Volute losses are also significant and they contribute 31% of total hydraulic losses.

This acknowledges the work Y.Senoo and H.Hayami [14], stating that 30% or more kinetic energy at diffuser exit remains unconverted to pressure energy. Leakage losses are found 15% of actual discharge, while mechanical losses are observed 12% of input power. More deviation of exit velocities is observed in small size impellers. It confirms that shorter blade passage height (r_2-r_1) will produce more slip. This is due to absence of proper flow guidance within blade passage. Blade height is important parameter and should be considered while designing.

These facts lead to the conclusion that impeller and volute losses are to be minimized to improve hydraulic efficiency of fans. Impeller losses can be reduced by adopting smaller diameter ratio (d_1/d_2) and tapered shroud plates to reduce rate of diffusion and increase slip factor. Vaned diffuser will help to reduce eddies in volute casing and hence reducing volute losses [1].

(6) Three dimensional CFD analysis results for centrifugal fan at steady and incompressible flow using MRF approach, gives good insight on flow behaviour. Energy transfer from impeller to fluid is seen by pressure and velocity contours within blade passage. Low and high pressure regions along suction and pressure side of a blade are visualized. Non-uniform pressure distributions are observed at off design conditions due to flow acceleration or retardation.

It is observed that average impeller inlet total pressure is 99 Pa and average impeller exit total pressure is 861 Pa. It makes 762 Pa impeller stage stagnation pressure rise. While in volute casing, total pressure drops from 861 Pa to 790 Pa. At nose radius plane, total pressure is 800 Pa at 20.56 m/s fluid velocity. Drop in total pressure indicates substantial volute losses and flow recirculation. These results follow standard fan performance H-Q curves given in literature [8].

Jet and wakes are observed in the vicinity of tongue region. The flow phenomenon of recirculation near tongue region is confirmed by numerical analysis as shown in stream line diagram given in figure number 4.3. It shows that design of tongue is very much important in fan design to reduce back flow and recirculation. Numerical simulation of all cases has reaffirmed the validity of proposed unified design methodology for radial tipped centrifugal fan.

Finally it may be stated that the numerically and experimentally validated unified design methodology, slip factors for radial tipped centrifugal fans, loss analysis and 3-D CFD flow simulation may be considered as major outcome of present work.